Bridge Analysis Using Finite Element Software

Sharmila K.L[#1], S. Selva Sajitha[#2]

[1]*Postgraduate Student,* [2]*Assistant Professor*

[#]*Department of Civil Engineering, Dr. Sivanthi Aditanar College of Engineering Tamil Nadu, India*

[1]sharmi.kl92@gmail.com, [2]ssajitha2008@gmail.com

***Abstract*— Bridges are defined as structures, which provide a connection or passage over gap without blocking the opening or passageway beneath. Various forces act on a bridge structure such as imposed load, wind load, forces due to buoyancy, seismic load, forces due to varying velocity of water, etc. This study was carried out to simulate the tensile damage occurring due to the seismic forces in the concrete bridge. The three dimensional model of the bridge is created on the basis of the geometrical data available. The seismic performance of the reinforced concrete bridge to moderate earthquake is assessed. The concrete damage plasticity model is assumed to represent the inelastic behaviour of the concrete material under dynamic loading. It resulted in tensile damage in some parts of the bridge. The results obtained in ABAQUS were in good agreement with those results reported by Joanna M. Dulinska. The use of ABAQUS is cost effective and time saving.**

***Keywords*— Seismic analysis, Concrete Damage Plasticity, Finite Element Analysis, Abaqus.**

I. Introduction

The finite element simulation of the solid elements includes creation of solid geometry, selection of element type, meshing and analysing for obtaining the results. The finite element method is general technique for obtaining the approximate solution for variety of engineering problems. The basic concept is that a body of structure may be discretized into smaller elements of finite elements are called finite element. The original body or structure is then considered as the assemblage of these elements connected at a finite number of joints called nodes or nodal joints.

Discretization of the structure or body into finite forms, the first step in the analysis of a complicated structural system. The properties of the element are formulated and combined to obtain the solution for the entire body of the structure. For example the displacement formulation is widely adopted in the finite element analysis, simple function known as shape function are chosen to approximate the variation of displacement of the nodes of the element.

The stresses and strains within the element will also be expressed in terms of nodal displacement. Then the principle virtual displacement of minimum potential energy is used to derive the equation of equilibrium for the element and the nodal displacement will be unknown in the equations. The equations of equilibrium for the entire structure or body are then obtained by combining the equilibrium equations of each element such that the continuity of displacement is ensured at each node where the elements are connected. The necessary boundary conditions are imposed and the equations of equilibrium are solved for the nodal displacement.

A survey of the effective finite element formulations for the analysis of the solid structures is presented. First the basic requirements for the solid elements are discussed. In which it is emphasized that generalized and reliability ear most important items. A general displacement based formulation is then briefly reviewed.

II. Finite Element Software

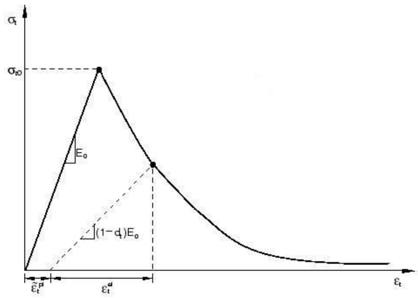
Abaqus FEA (formerly ABAQUS) is a software suite for finite element analysis and computer aided engineering, originally released in 1978. The Finite element method (FEM) is a powerful technique originally developed for numerical solution of complex problems in structural mechanics, and it remains the method of choice for complex systems. In the FEM, the structural system is modelled by a set of appropriate finite elements interconnected at points called nodes. Elements may have physical properties such as thickness, coefficient of thermal expansion, density, young's modulus, shear modulus and Poisson's ratio. The Abaqus Unified FEA product suite offers powerful and complete solutions for both routine and sophisticated engineering problems covering a vast spectrum of industrial applications. Abaqus is used in the automative, aerospace, and industrial products industries. The product is popular with academic and research institutions due to the wide material modelling capability, and the program's ability to be customized. Abaqus also provides a good collection of multiphysics capabilities, such as coupled acoustic-structural, piezoelectric, and structural-pore capabilities, making it attractive for production-level simulations where multiple fields need to be coupled. Abaqus was initially designed to address non-linear physical behaviour; as a result, the package has an extensive range of material models such as elastomeric (rubber like) material capabilities.

III. CDP Model

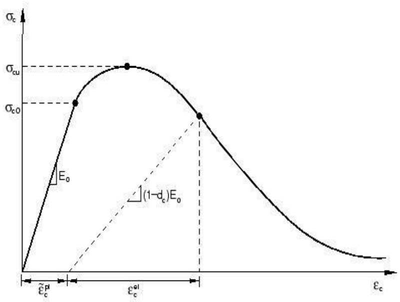
The concrete damaged plasticity(CDP) model in Abaqus provides a general capability for modelling concrete and other quasi-brittle materials in all types of structures (beams, trusses, shells, and solids). It uses concepts of isotropic damaged elasticity in combination with isotropic tensile and compressive plasticity to represent the inelastic behaviour of concrete. Further this can be used for plain concrete, even though it is intended primarily for the analysis of reinforced concrete structures and can be used with rebar to model concrete reinforcement. The concrete damaged plasticity model is designed for applications in which concrete is subjected to monotonic, cyclic, and/or dynamic loading under low confining pressures. It consists of the combination of non associated multi-hardening plasticity and scalar (isotropic) damaged elasticity to describe the irreversible damage that occurs during the fracturing process. It can be defined to be sensitive to the rate of straining.

The concrete damaged plasticity model can be used in combination with material damping. If stiffness proportional damping is specified, Abaqus calculates the damping stress based on the undamaged elastic stiffness. This may introduce large artificial damping forces on elements undergoing severe damage at high strain rates.

The model assumes that the uniaxial tensile and compressive response of concrete is characterized by damaged plasticity as shown in figure below.



(a)



(b)

Fig 1 (a) Tension, (b) Compression

Under uniaxial tension the stress-strain response follows a linear elastic relationship until the value of the failure stress, σ_{t0}, is reached. The failure stress corresponds to the onset of micro-cracking in the concrete material. Beyond the failure stress the formation of micro-cracks is represented macroscopically with a softening stress-strain response, which induces strain localization in the concrete structure. Under uniaxial compression the response is linear until the value of initial yield, σ_{c0}. In the plastic regime the response is typically characterized by stress hardening followed by strain softening beyond the ultimate stress, σ_{cu}. This representation, although somewhat simplified, captures the main features of the response of concrete.

The effective stress is defined as $\bar{\boldsymbol{\sigma}} = \mathbf{D}_0^{el} : (\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}^{pl})$.

The plastic flow potential function and the yield surface make use of two stress invariants of the effective stress tensor, namely the hydrostatic pressure stress,

$\bar{p} = -\frac{1}{3}\text{trace}\,(\bar{\boldsymbol{\sigma}})$,

and the Mises equivalent effective stress,

$\bar{q} = \sqrt{\frac{3}{2}(\bar{\mathbf{S}} : \bar{\mathbf{S}})}$,

where $\bar{\mathbf{S}}$ is the effective stress deviator, defined as

$\bar{\mathbf{S}} = \bar{\boldsymbol{\sigma}} + \bar{p}\mathbf{I}$.

IV. ANALYTICAL STUDY

In this paper, the assessment of seismic performance of the reinforced concrete bridge to the moderate earthquake was analytically presented. In order to represent the inelastic behaviour of the concrete material under dynamic loading the concrete damage plasticity model was assumed. The analysis proved that the first natural frequency of the bridge lied in the range of dominant frequencies of the shock that caused the amplification of the dynamic response. It turned out that the tensile damage (cracking) appeared in some parts of the bridge and the stiffness of the concrete material was degraded even under moderate seismic event. The dynamic response of the bridge to the moderate seismic shock was calculated by time history analysis. A small step increment of 0.001 s was introduced for this highly nonlinear analysis to obtain convergence. The evolution of tensile damage and elastic stiffness degradation of the middle bottom part of the bridge deck was clearly observed.

A. Dimensions of the Bridge Structure

The length of the central span was 26.1 m. The three piers of 2.95 m high with a diameter of 1.1 m were located regularly at a distance of 3.75 m. The abutments were situated 7.05 m away from the extreme piers. The fixed boundary conditions reflected the high rigidity of the foundation rock. The bridge was equipped with elastomeric bearings as linking elements between the deck of the bridge and piers. The cross sectional area of the bearing was determined by the allowable pressure on the bearing support. The length and the width were both assumed 0.6 m. The bearing was composed of two steel cover plates, each 20 mm thick, between which elastomeric laminae reinforced with steel shims were placed. The thickness of one elastomeric layer was assumed 8 mm; the thickness of one steel reinforcement layer was 3 mm.

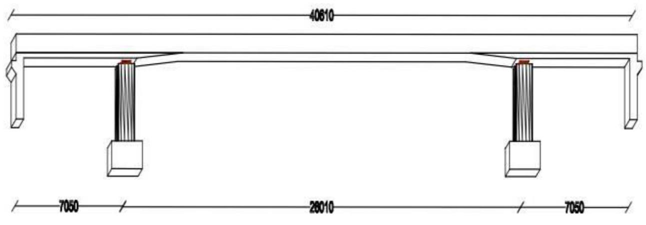


Fig. 2 Main Dimensions of the Bridge

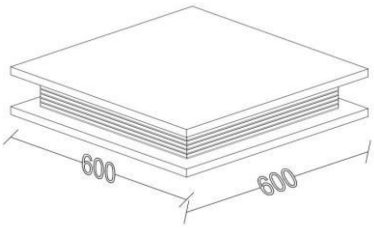


Fig. 3 Main Dimensions of the Elastomeric bearing

B. Material Properties

The elastic modulus for concrete was taken as 19.7GPa.Poisson's ratio for concrete was considered to be 0.19.The equivalent elasticity modulus was taken as 2.814 MPa. The Poisson's ratio of the elastomeric bearing was taken as 0.49. For the steel shims the elasticity modulus of 210 GPa and Poisson's ratio of 0.3 were considered.

C. Finite Element Simulation

The analytical results of the concrete bridge in the journal is verified. In the ABAQUS software, the concrete bridge is modelled as three dimensional deformable parts with concrete and the elastomeric bearings as solid elements. The concrete bridge is modelled for the concrete damage plasticity properties.

1) Properties:

To perform the analytical study, it is necessary to input the material properties such as modulus of elasticity E and Poisson's ratio for the material model of concrete. For this study, the following material properties of concrete and the elastomeric bearing in accordance with the literature.

2) *Loads and Boundary Conditions:*

In this study, the boundary conditions are assumed to be restrained in all directions i.e. U1, U2, U3 and the rotations Ur1, Ur2, Ur3 are all restrained and this condition is applied to the base of the pier and the abutment. The load is applied in the middle of the deck slab which will result in the tensile damage at the bottom of the slab.

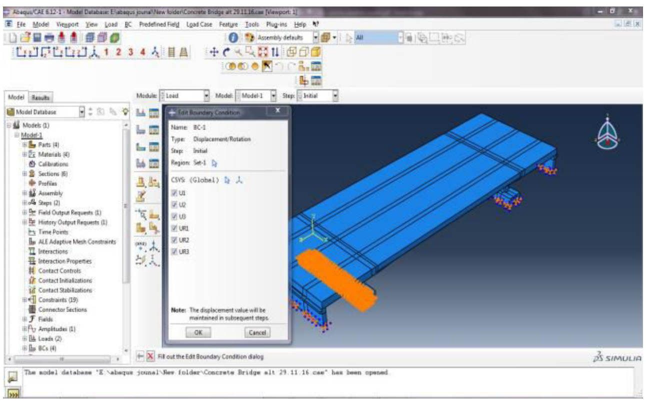


Fig. 4 Boundary Condition Used in FEM Analysis

3) *Elements Chosen*:

The element chosen for the concrete is the eight noded solid element as C3D8R. The geometry and the nodal location of the element are shown in figure 5.

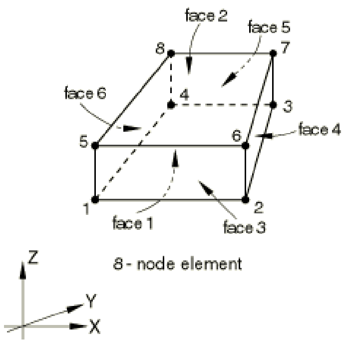


Fig. 5 C3D8R Element

It is the linear brick first order element which is of reduced integration to improve their bending behaviour. These elements are somewhat more expensive than the regular first-order displacement elements; however, they are significantly more economical than second-order elements. The reduced integration elements use reduced integration and, thus, have hourglass control modes. This element can be composed of a single homogeneous material and also can include several layers of different materials. For this analysis, it is considered as a single homogeneous material.

The element chosen for the elastomeric bearing is the ten noded continuum element namely C3D10M. The geometry and thee nodal location of the element are shown in figure 6.

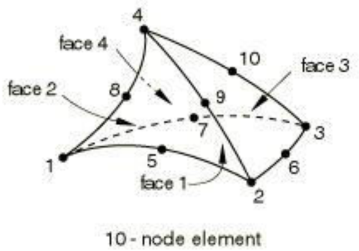


Fig. 6 C3D10M Element

It is a linear brick element which is a modified tetrahedron. This element is considered to be the Stress/Displacement element. For this analysis, it is considered as a single homogeneous material.

D. Analytical Results

Under the dead load, the maximum deflection of the slab takes place at the centre of the slab. The damage takes place due to the tension at the bottom of the slab at centre. The figure 6 shows the tension damage under dead load only.

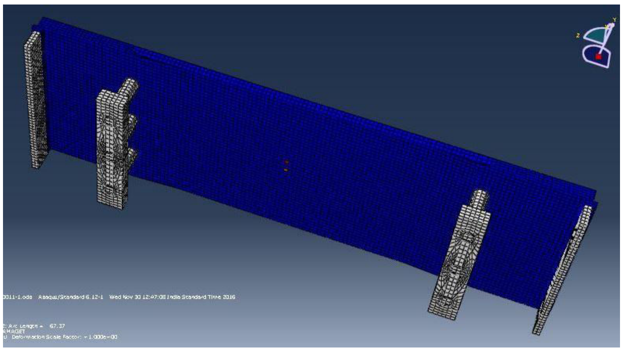


Fig. 7 Tension Damage of Slab under Dead Load from ABAQUS

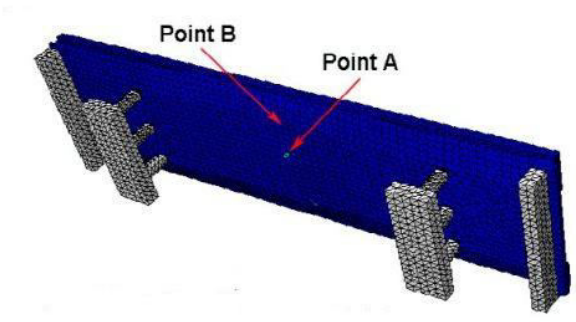


Fig. 8 Tension Damage of Slab under Dead Load from Journal

The maximum strain occurs at the bottom of the slab at the centre where the tension damage starts to occur in the slab. The analysis of the bridge under load before the seismic activity i.e. at t= 0s shows approximately similar results as in the journal.

The strain curve plotted against time in the seismic loading up to t= 20s is shown in figure 9. The figure clearly explains the clear correlation of the ABAQUS results with the analytical results in the literature.

The graph is plotted for the values of Equivalent Plastic Strain available in the journal and the analysis results from ABAQUS software in the y-axis and Time in the x- axis.

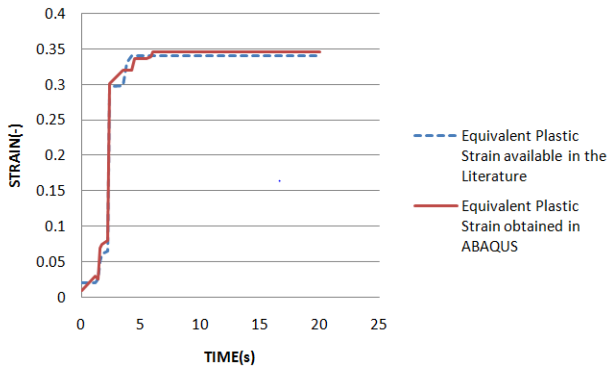


Fig. 9 Time Vs Strain Graph

V. Conclusion

The concrete bridge was modeled using ABAQUS standard software in which elements of type solid continuum elements C3D8R were chosen. It was found that this element represents the behaviour of concrete very well. It is proved that elements of type solid continuum elements C3D8R (ABAQUS) give approximately similar results for the geometrically non-linear large-deformation analysis. The finite element analysis using ABAQUS shows good agreement with the analytical results published in the literature. So simulation of bridge structures can be done using the ABAQUS software and it proves to be cost effective too.

References

[1] Abaqus Analysis User's Manual, Version 6.12.3, SIMULIA.

[2] Joanna M. Dulinska, Radoslaw Szczerba, "Assessment of Concrete Bridge Performance under Moderate Seismic Shock Using Concrete Damage Plasticity Model", *Procedia Engineering,* 57, 1319 – 1328.

[3] Hao Qin, Xianzhong Zhao, Yiyi Chen, Bin Wang, "Elasto-Plastic Time History Analysis of Shanghai Tower Under Rare Earthquake", CTBUH Research Paper,2011.

[4] Nina Zheng, Jing Zhou, Yuanyuan Yin, Jun Han, Shuyan Ji,"Nonlinear Time History Analysis of Low Masonry Structure with Tie-Columns", WCEE LISBOA,2012.

[5] Sam Lee, "Nonlinear Dynamic Earthquake Analysis of Skyscrapers", CTBUH Research Paper,2008.

[6] Selcuk Saatci, Frank J. Vecchio,"Nonlinear Finite Element Modeling of Reinforced Concrete Structures under Impact Loads",ACI Structural Journal,Title no. 106-S67,September-October2009.