ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

IJESRT

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

STRUCTURAL ANALYSIS AND OPTIMIZATION OF BRIDGE PIER USING ANSYS

Premsai.T
Structural Engineering, Csi College Of Engineering, India

ABSTRACT

The main objective of this study is to explore the analysis of a bridge pier. This has entailed performing a detailed static analysis. The study deals with static structural analysis. A proper Finite Element Model is developed using Cad software Pro/E Wildfire 5.0. In this project we are doing the decrement of width of bridge pier. This project we are designed a 3D model of the bridge pier by using pro-E software and the analysis is taken by different width values of the bridge pier and simulation work is done in the ANSYS software. The study of the bridge pier is to know the variation of displacement, stresses, Quantity of steel and Quantity of concrete.

Keywords: Pier, 3D Modeling, Meshing, Static Analysis, Maximum Allowable Deflection.

INTRODUCTION

Bridges are lifeline facilities that must remain functional even after major earthquake shaking; their damage and collapse may not only cause loss of life and property, but also hamper post-earthquake relief and restoration activities. In some major earthquakes in the past, a large number of bridges suffered damages and collapsed due to failure of foundation (structural and geotechnical), substructure, superstructure, and superstructure-substructure and substructure-foundation connections.

Bridge foundation is not easily accessible for inspection and retrofitting after an earthquake, and any inelastic action or failure of the superstructure renders the bridge dysfunctional for a long period. Connection failure is generally brittle in nature and hence avoided. Therefore, the substructure is the only component where inelasticity can be allowed to dissipate the input seismic energy and that too in flexural action. In addition, a flexural damaged pier can be more easily retrofitted.

MATERIALS AND METHODS

Failure of bridge piers

During recent earthquakes, such as the Great Hanshin Earthquake in Japan, January 17, 1995, many of the reinforced concrete piers or columns of highway and railway bridges suffered mainly severe diagonal shear failure in addition to other features of damages. The recognized reason for such severe collapse was that the piers were designed in such a way that they were provided with insufficient quantities of shear or lateral reinforcement which resulted in non-ductile behavior of the piers during the motion. Such severe collapse makes us to more deeply recognize that many of our RC bridges did not have sufficient strength and ductility to resist strong ground shaking. It is very important to analyze dynamic failure mechanisms and to investigate the causes of the severe shear failure. An accurate estimation of ductility levels is also necessary. As demonstrated in the previous studies (Machida and Mutsuyoshi 1988, Okamura and Maekawa 1991, Maekawa and Shawky 1997 and Bathe 1996), shear reinforcement has significant role on final failure mode of RC piers. Shear reinforcement shares both of concrete and longitudinal reinforcement in resisting shear stresses and thus increases shear strength of piers. Also, shear reinforcement increases ultimate strength and the corresponding strain due to confinement, increases ductility level of piers and prevents buckling of longitudinal bars. In the previous studies, the effect of shear reinforcement on failure mode of piers was determined either experimentally or theoretically based on two-dimensional analysis, which is not accurate enough for such structures. Consequently, it is required to evaluate accurately the limits of the quantities of shear reinforcement, which are needed to prevent or to reduce the occurrence of diagonal failure.

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

METHODOLOGY

In this work, an attempt is made to decrease the weight of the bridge pier without exceeding allowable deflection value. So the quantity of steel and concrete can be minimized to reduce the overall cost of the bridge pier construction at the same while to ensure the strength and life time of bridge. Three dimensional model of bridge pier is prepared in PRO E. Static structural analysis of bridge pier models is performed in ANSYS workbench 14.5.

Planning

Planning is a foremost step in methodology. To identify all the information and requirement of software, planning must be done in the proper manner to avoid design failure. The planning phase has two main elements namely data collection and the software requirements.

Data collection

Data collection is a stage in any area of study. At this stage I planned about the projects resources and requirements, literature studies and schedule to get more information in this study. All the materials are collected from journal, texts book and research papers gathered from libraries and Internet. Within the data collection period I have found the study about the design of bridge, Solid Mechanics in the Internet and do some research about the project related.

Software Requirements

For software requirement, I have chosen Auto-CAD, PRO-E and ANSYS WORKBENCH Software. Auto-CAD is a vital software package for building design and special tool for design elevation. PRO E is a software tool suite used primarily for three dimensional designs and detailing. By using this software we can create only 3D modeling and it is converted into IGES format which is applicable to import into ANSYS software. The next software is ansys workbench which is widely used for finite element analysis purpose. By using this software we can get the result value of maximum deformation, strain and stress values of bridge pier.

Materials Used For Bridge Pier

Two materials are involved in the bridge structures in this study; i.e., concrete and steel rod. Generally Concrete has high compressive strength, but much lower tensile strength. Load enduring capacity Depends up on Concrete mixture. It is usually reinforced with materials that are strong in tension (often steel). Steel is an alloy of iron and carbon that is widely used in construction of roads, railways, other infrastructure, appliances and buildings. Because of its high tensile strength and low cost.

Checking

After the analysis of bridge pier is finished, I need to check the result values to ensure the bridge pier is in safest condition. At the same time, we are going to compare the results on bridge pier under static loading condition in different dimensional structure in order to improve the existing design and possibly reduce cost in bridge pier construction without reducing strength of bridge pier.

Methodology

- Planning about the whole process
- Required data has been collected from books and journals
- 3d modelling of the bridge pier using pro e wildfire 5.0 software
- Pro e model imported into the ansys workbench 14.5
- Meshing of model carried out by finite element method
- By static analysis different width of bridge piers are analyzed
- Then the results are tabulated to find the maximum allowble deflection limit of pier

PRO-E

***Unit Based Attributes* -** This is the ability to fully leverage Pro/ENGINEER's parameters with units in conjunction with Windchill's unit based attribute capabilities. By allowing the mapping of these attribute types you can convey units of measure for specific attributes between the two systems when a Pro/ENGINEER parameter is designated and the Windchill type definition includes the same attribute assigned a compatible unit of measure.

This means if a parameter in Pro/ENGINEER representing a diameter is also assigned a unit of measure, both the value and the unit of measure are conveyed to Windchill thus providing additional context to the attribute value within the data management system

EXTRUDE: This command is used to create the material (to make 3D object from 2D sketch) from the sketched entities. The entities may be circle, line or rectangle, etc. Select the extrude icon from the right tool chest then select the sketched part in the window, enter the extrude length and press the middle mouse button to finish the extrude command. There is a provision for removing material in pro –e which is called cut. The main condition to create the solid model is the sketched section must be closed.

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

Fig. 1 Isometric view of Bridge Pier

ANSYS

ANSYS is an engineering simulation software provider founded by software engineer John Swanson. It develops general-purpose finite element analysis and computational fluid dynamics software. While ANSYS has developed a range of computer-aided engineering (CAE) products, it is perhaps best known for its ANSYS Mechanical and ANSYS Multiphysics products. ANSYS Mechanical and ANSYS Multiphysics software are non exportable analysis tools incorporating pre-processing (geometry creation, meshing), solver and post-processing modules in a graphical user interface. These are general-purpose finite element modeling packages for numerically solving mechanical problems, including static/dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems.

ANSYS Mechanical technology incorporates both structural and material non-linearity. ANSYS Multiphysics software includes solvers for thermal, structural, CFD, electromagnetic, and acoustics and can sometimes couple these separate physics together in order to address multidisciplinary applications. ANSYS software can also be used in civil engineering, electrical engineering, physics and chemistry. ANSYS, Inc. acquired the CFX computational fluid dynamics code in 2003 and Fluent, Inc. in 2006. The CFD packages from ANSYS are used for engineering simulations. In 2008, ANSYS acquired Ansoft Corporation, a leading developer of high-performance electronic design automation (EDA) software, and added a suite of products designed to simulate high-performance electronics designs found in mobile communication and Internet devices, broadband networking components and systems, integrated circuits, printed circuit boards, and electromechanical systems. The acquisition allowed ANSYS to address the continuing convergence of the mechanical and electrical worlds across a whole range of industry sectors.

FINITE ELEMENT ANALYSIS

FEA consists of a computer model of a material or design that is stressed and analyzed for specific results. It is used in new product design, and existing product refinement. A company is able to verify a proposed design will be able to perform to the client's specifications prior to manufacturing or construction. Modifying an existing product or structure is utilized to qualify the product or structure for a new service condition. There are generally two types of analysis that are used in industry: 2-D modeling, and 3-D modeling. While 2-D modeling conserves simplicity and allows the analysis to be run on a relatively normal computer, it tends to yield less accurate results. 3-D modeling, however, produces more accurate results while sacrificing the ability to run on all but the fastest computers effectively. Within each of these modeling schemes, the programmer can insert numerous algorithms (functions) which may make the system behave linearly or non-linearly. Linear systems are far less complex and generally do not take into account plastic deformation. Non-linear systems do account for plastic deformation, and many also are capable of testing a material all the way to fracture.

Finite Element Analysis (FEA) was first developed in 1943 by R. Courant, who utilized the Ritz method of numerical analysis and minimization of variational calculus to obtain approximate solutions to vibration systems. Shortly thereafter, a paper published in 1956 by M. J. Turner, R. W. Clough, H. C. Martin, and L. J. Topp established a broader definition of numerical analysis. The paper centered on the "stiffness and deflection of complex structures". By the early 70's, FEA was limited to expensive mainframe computers generally owned by the aeronautics, automotive, defense, and nuclear industries. Since the rapid decline in the cost of computers and the phenomenal increase in computing power, FEA has been developed to an incredible precision. Present day supercomputers are now able to produce accurate results for all kinds of parameters. FEA consists of a computer model of a material or design that is stressed and analyzed for specific results. It is used in new product design, and existing product refinement. A company is able to verify a proposed design will be able to perform to the client's specifications prior to manufacturing or construction. Modifying an existing product or structure is utilized to qualify the product or structure for a new service condition. In case of structural failure, FEA may be used to help

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

determine the design modifications to meet the new condition. There are generally two types of analysis that are used in industry: 2-D modeling, and 3-D modeling. While 2-D modeling conserves simplicity and allows the analysis to be run on a relatively normal computer, it tends to yield less accurate results. 3-D modeling, however, produces more accurate results while sacrificing the ability to run on all but the fastest computers effectively. Within each of these modeling schemes, the programmer can insert numerous algorithms (functions) which may make the system behave linearly or non-linearly. Linear systems are far less complex and generally do not take into account plastic deformation. Non-linear systems do account for plastic deformation, and many also are capable of testing a material all the way to fracture.

STRUCTURAL ANALYSIS

A static analysis calculates the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time-varying loads. A static analysis can, however, include steady inertia loads (such as gravity and rotational velocity), and time-varying loads that can be approximated as static equivalent loads (such as the static equivalent wind and seismic loads commonly defined in many building codes).

Element is an entity, into which a system under study can bedivided into. An element definition can be specified by nodes. The shape(area, length, and volume) of the element depends upon the nodes with which it is made up of.

Nodes are the corner points of the element. Nodes are independent entities in the space. These are similar to points in geometry. By moving a node in space an element shape can be changed**.**

0-D Element has the shape of the point, it requires only one node to define it.

1-D Element has the shape of the line/curve and hence requires minimum of two nodes to define it.

2-D Element is an area element, which has the shape of the quadrilateral/triangle and hence requires minimum four/three nodes to define it.

3-D Elements is a volume element, can take the shape of a Hexahedron or a Wedge or a Tetrahedron. Hexahedron element requires 8 nodes to define its shape. A Pent element requires 6 nodes to define its shape. Similarly 4 nodes are required to define a Tetra element. The element is said to be linear or 1storder when it doesn't have any mid side nodes. If the mid side nodes are present then those elements are called Quadratic or 2ndorder elements. For linear elements the edge is defined by a linear function called shape function whose degree is one. For the elements having mid side nodes on the edge quadratic function called shape function whose degree is two is used. The higher order elements when over lapped on geometry can represent complex shapes very well within few elements. Also the solution accuracy more with the higher order elements.

PROCEDURE FOR ANSYS ANALYSIS

Static analysis is used to determine the displacements stresses, stains and forces in structures or components due to loads that do not induce significant inertia and damping effects. Steady loading in response conditions are assumed. The kinds of loading that can be applied in a static analysis include externally applied forces and pressures, steady state inertial forces such as gravity or rotational velocity imposed (non-zero) displacements, temperatures (for thermal strain).

1. Preparatory work prior to analysis.

2. Preprocessor modeling through the preprocessor: Defining: define cell types, constants, material properties, etc. Modeling: create model in work page; or build directly with external software, then import. Meshing: (to recommend smart grid divisions). Checking: check that the model is correct before saving. (Note: because ANSYS has no return key, it is recommended that the Save As command be used after the implementation of each of these key steps. This will prevent errors in the modeling or solving processes, and facilitate the call)

3. Solving by Solution: Select the type of analysis; set analysis options. Apply additional load and constraints. Set load step options. Solving: (the default solver option can be used, if the load step has been set to select for solving, according to the load step options-solver).

4. Viewing results via General Postprocessor: After solving, review the model's analysis of data, deformation maps, stress map, displacement map, etc.

LOADS IN A STRUCTUL ANALYSIS

Static analysis is used to determine the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed; that is, the loads and the structure's response are assumed to vary slowly with respect to time.

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

ANALYSING THE BRIDGE PIER

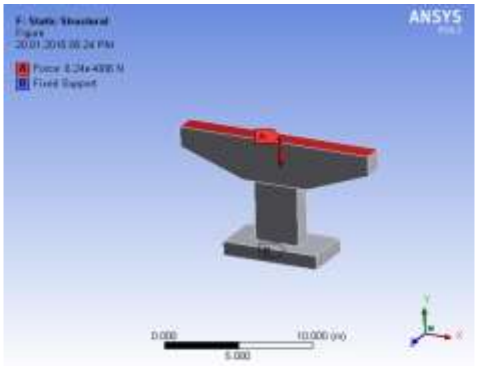


Fig. 5 Input Values applied on the Bridge Pier in ANSYS Workbench

RESULTS FOR BRIDGE PIER

Deformation in continuum mechanics is the transformation of a body from a reference configuration to a current configuration. A configuration is a set containing the positions of all particles of the body. As deformation occurs, internal inter-molecular forces arise that oppose the applied force. If the applied force is not too great these forces may be sufficient to completely resist the applied force and allow the object to assume a new equilibrium state and to return to its original state when the load is removed. A larger applied force may lead to a permanent deformation of the object or even to its structural failure. So the deformation value should in allowable limit.

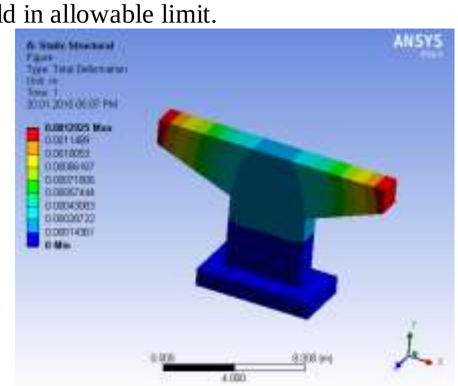


Fig.6 Actual Design (Total Deformation)

EQUIVALENT STRESS

When a material is loaded with a force, it produces a stress, which then causes a material to deform.

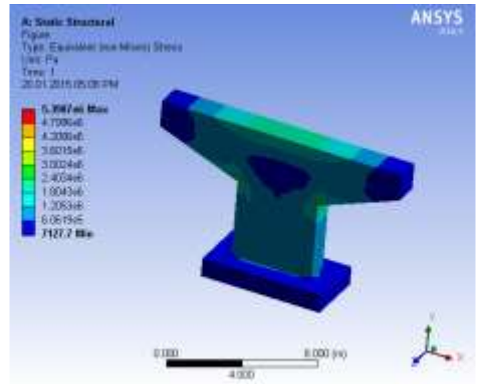


Fig. 7 Equivalent Stress

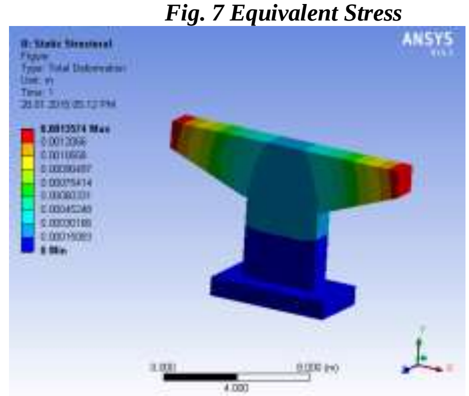


Fig. 8 Modified Design 1 (Total Deformation)

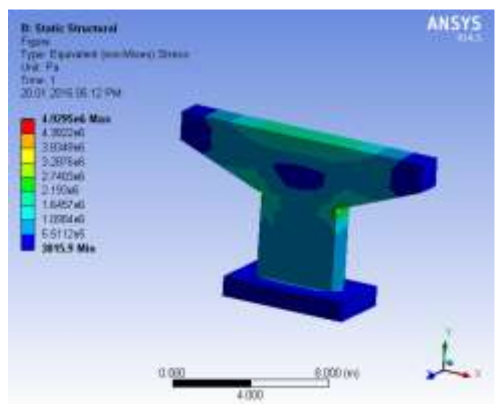


Fig. 9 Equivalent Stress

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

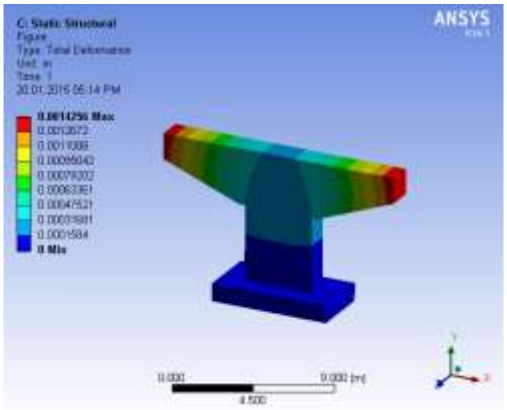


Fig. 10 Modified Design 2 (Total Deformation)

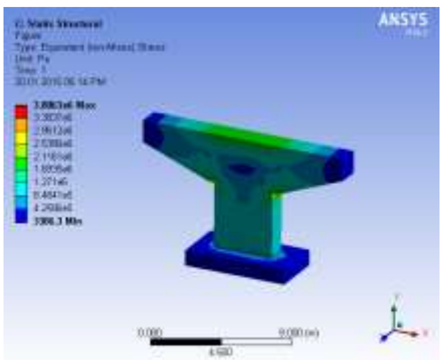


Fig. 11 Equivalent Stress

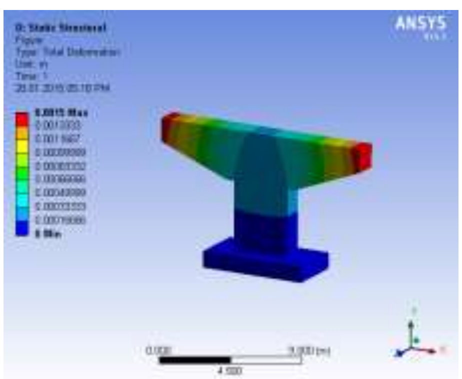


Fig. 12 Modified Design 3 (Total Deformation)

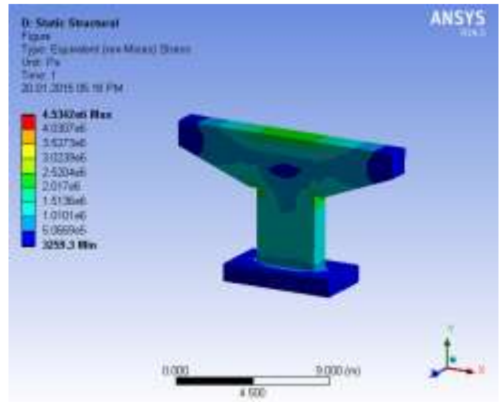


Fig. 13 Equivalent Stress

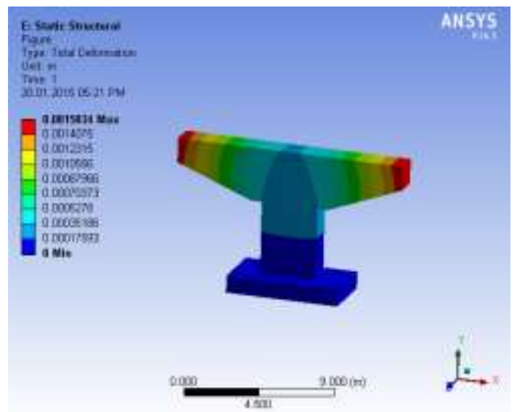


Fig. 14 Modified Design 4 (Total Deformation)

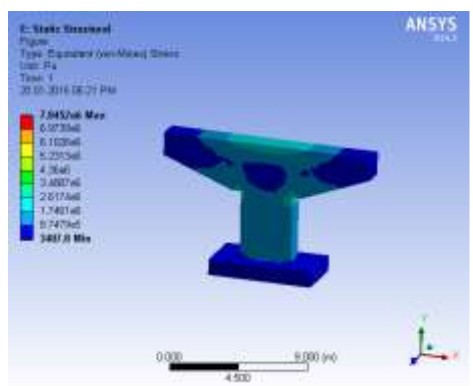


Fig. 15 Equivalent Stress

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

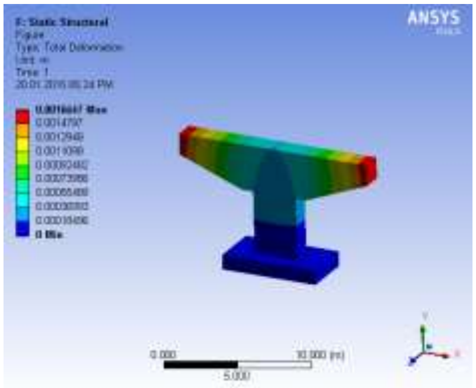


Fig. 16 Modified Design 5 (Total Deformation)

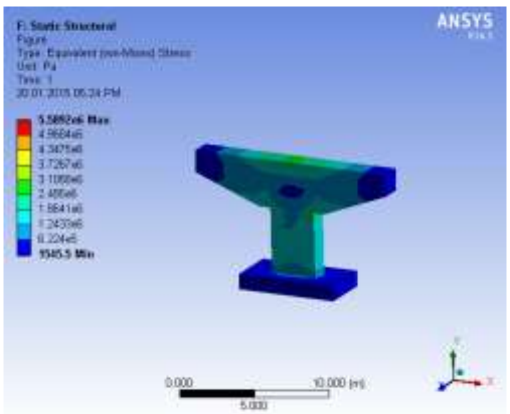


Fig. 17 Equivalent Stress

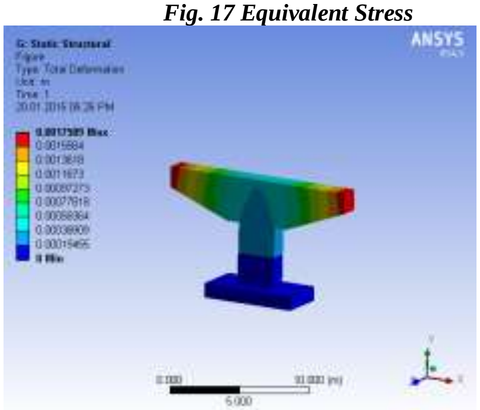


Fig. 18 Modified Design 6 (Total Deformation)

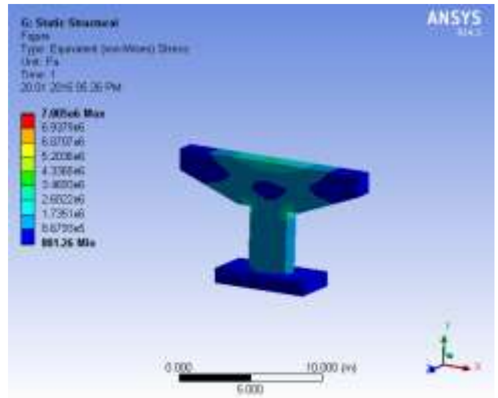


Fig. 19 Equivalent Stress

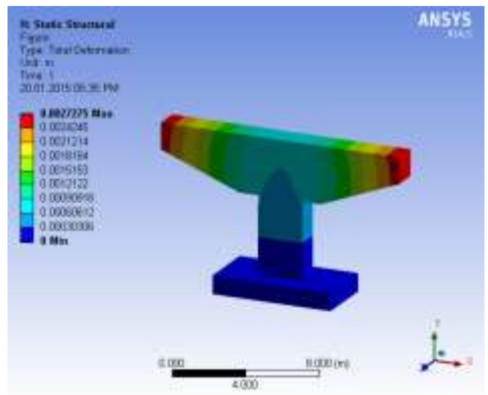


Fig. 20 Modified Design 7 (Total Deformation)

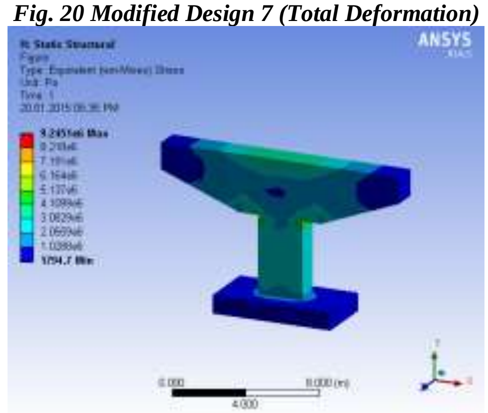


Fig. 21Equivalent Stress

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

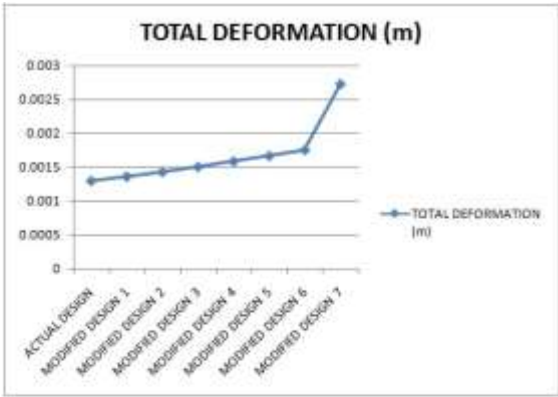


Table 1 Total Deformation (m)

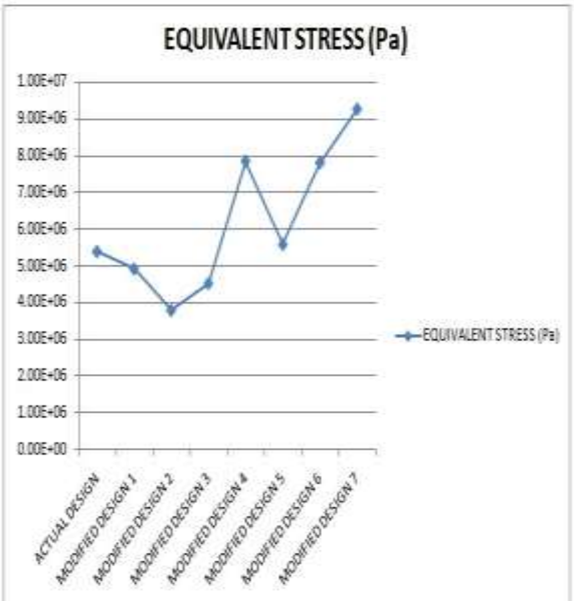
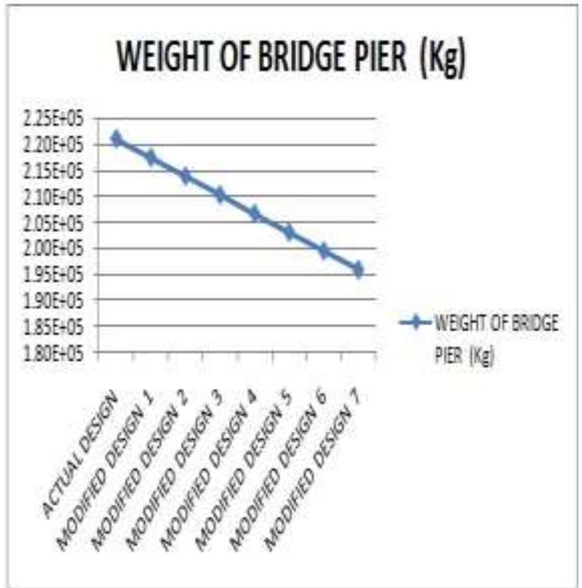


Table 2 Equivalent Stress (Pa)



ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

Table 3 Weight of Bridge Pier (Kg)

COMPARISON TABLE

	TOTAL DEFORMATION (m)	EQUIVALENT STRESS (Pa)
ACTUAL DESIGN	0.001292	5.398e6
MODIFIED DESIGN 1	0.001357	4.929e6
MODIFIED DESIGN 2	0.001425	3.806e6
MODIFIED DESIGN 3	0.00150	4.534e6
MODIFIED DESIGN 4	0.001583	7.845e6
MODIFIED DESIGN 5	0.001664	5.589e6
MODIFIED DESIGN 6	0.00175	7.805e6
MODIFIED DESIGN 7	0.002727	9.254e6

WEIGHT OF BRIDGE PIER

	WEIGHT OF BRIDGE PIER (Kg)
ACTUAL DESIGN	2.2111e5
MODIFIED DESIGN 1	2.1754e5
MODIFIED DESIGN 2	2.1396e5
MODIFIED DESIGN 3	2.1039e5
MODIFIED DESIGN 4	2.0682e5
MODIFIED DESIGN 5	2.0324e5
MODIFIED DESIGN 6	1.9967e5
MODIFIED DESIGN 7	1.9609e5

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

CONCLUSION Analysis results of the bridge pier with reduction of width values are listed in the Table. Analysis has been carried out from actual design to seven modified designs. The results such as total deformation, equivalent elastic strain and equivalent stress for each modal of bridge pier from actual design to seven modified designs are determined. Comparing the results upto the sixth modified design has allowable total deformation value. Seventh modified design exceeds its allowable deformation and equivalent stress limit, so the analysis has been stopped here. The amount of concrete and steel minimized by 9.696% while using sixth modified design bridge pier. Hence it is concluded that sixth modified is suitable for bridge pier. While carrying out this project we are able to study about the 3Dmodelling software (PRO-E) and Study about the analyzing software (ansys) to develop our basic knowledge to know about the structural design. This fragment should obviously state the foremost conclusions of the exploration and give a coherent explanation of their significance and consequence.

ACKNOWLEDGEMENTS

I would like to express my philosophical sense of deepest gratitude to
Mr.Sam Chelladurai,M.E.,(Ph. D.),
Head of the Department of Structural Engineering, for his treasured suggestions, encouragement and his valuable guidance.

REFERENCES

[1] Rashmi R.Vanahalli, Dr. S.V.Itti developed an article regarding to “study on RCC bridge pier using ansys” in August 2013 on International Journal of Engineering and Innovative Technology.

[2] Rupen Goswami and C. V. R. Murty present a review of “seismic strength design provisions for reinforced concrete (RC) bridge piers" given in Indian codes in 2002.

[3] Chung-Wei Feng, Shen-Haw Ju, Hsun-Yi Huang, and Pao-Sheng Chang discussed “using genetic algorithms to estimate the scour depth around the bridge pier” **.**

[4] Fabio F. Taucer, Carlo Paulotto, Gustavo Ayala develops “Hollow bridge-pier properties for response spectrum analysis” in May 2010.

[5] The dissertation entitled “Seismic Analysis of Existing Bridges with Detailing Deficiencies”, by Davide Kurmann on February, 2009.

[6] D.S. Wang, Q.H. Ai, H.N. Li, B.J. Si and Z.G. Sun presents “Displacement based seismic design of RC bridge piers: Method and experimental evaluation” in oct 2008 on The 14th World Conference on Earthquake Engineering .

[7] Hayder Ala’a Hasan, Ammar A. Abdul Rahman and Hani Aziz Ameen, “Finite element analysis of reinforced concrete bridge pier subjected to seismic loading”, American Journal of Science and Industrial Research, 2012.

[8] Gary R. Consolazio, A.M. and David R. Cowan, “Numerically Efficient Dynamic Analysis of Barge Collisions with Bridge Piers”, 2004.

[9] MiaMF, NagoH2003 Design method of time-dependent local scour at circular bridge pier. J. Hydraul.

[10] Goswami, R., Investigation of Seismic Shear Design Provisions of IRC Code for RC Bridge Piers Using Displacement-Based Pushover Analysis, Master of Technology Thesis, 2002. Department of Civil Engineering, Indian Institute of Technology Kanpur, India.

[11] Jain, S.K., and Murty, C.V.R., “A state of-the-art review on seismic design of bridges – CALTRANS, TNZ and Indian codes,” The Indian Concrete Journal, 1998.

[12] Richardson, J.E. and Panchang, V.G. (1998), “Three- Dimensional Simulation of Scour-Inducing Flow at Bridge Piers”, Journal of Hydraulic Engineering, Johnson, P.A. and Dock, D.A., “Probabilistic Bridge Scour Estimates”, Journal of Hydraulic Engineering, 1998.

[13] Bateni, S.M. and Borghei, D.-S. Jeng., “Neural network and neuro-fuzzy assessments for scour depth around bridge piers”, Engineering Applications of Artificial Intelligence, 2007.

[14] Mahmut Firat, and Mahmud Gungor., “Generalized Regression Neural Networks and Feed Forward Neural Networks for prediction of scour depth around bridge piers”, Advances in Engineering Software, 2009.

[15] Melville, B.W. and Raudkivi, A.J., “Effects of Foundation Geometry on Bridge Pier Scour”, Journal of Hydraulic Engineering, 1996.