Finite Element Analysis of Bridge Riser for Automobile Application Robots

J.Niresh[1*] Subramanian M[2] Neelakrishnan S[3]
{nireshcbe@gmail.com[1], sriisubbumani@gmail.com[2],snk.auto@psgtech.ac.in[3]}

[1*]Assistant Professor, [2]Associate Professor, [3]Professor & Head
Department of Automobile Engineering PSG College of Technology, Coimbatore, Tamil Nadu, India

Abstract. In recent days industrial robots having having exceptionally imperative part, from material handling to assembling of parts in vehicle. We are reliable on robots. Mounting of robots to perform its required tasks effectively under various loading conditions within the time .If the structural base frame is overdesigned or under designed then many types of problem can take place like, vibrations, fatigue failures, high instability, more material consumption, higher fabrication cost. The analysis is done by using ANSYS Workbench 2019 R2. Firstly make 3D geometry of robot platform as per inputs from industry. After applying boundary conditions like payload of robot, strengths connected on its body as well as minutes on the Bridge riser. Then the von-mises stress and total deformation of Bridge riser were obtained. Finally after getting results we will compare with conceptual design and decide the robot riser design is safe or not.

Keywords: Robot Bridge Riser, Robot Pedestals, Study Base, Von Mises Stress, Total Deformation.

1 Introduction

Robot could be a Self-operated machine which works like human or utilized to decrease endeavors of human being. Robots too do family work as well as mechanical work with incredible accuracy. In cutting edge days for mass generation in fabricating businesses we are utilizing different robots; automated work is exact as well as solid since of that we are utilizing different sum robots for mass generation.

When there's draw out utilize of robots and their parts at that point there are chance of disappointment happens in robot or there can harm in mechanical parts after some years. There's ought to plan the mechanical parts recently, dissect such like that execution is more noteworthy than that of the parts and stretch or stack capacity is more than the past one. Which comes about in expanding life cycle of robots and there's expanded benefit to businesses.

While using continuous robot there is damage is robotic parts or because of continuous vibration there is failure Occurs in Bridge riser. We are analysing the failure in Bridge riser. Which type of failure occurs, at which load, after how many cycles of robot. After that firstly we make design of robot by using NX CAD. And the file should be imported in ANSYS Workbench 2019 R2 for analysing . After model analysis we will find out von-misses stresses and total deformation of robot pedestal. Finally after results analysis we will finalise where the pedestal is safe for given loads as well as moments.

ICCAP 2021, December 07-08, Chennai, India

DOI 10.4108/eai.7-12-2021.2314480

Xiaoping Liao et al (2010), the investigation programming ANSYS discharge 10.0 have been utilized in the modular examination of the base ofwelding robot, and the regular frequencies and mode states of the initial ten requests have been processed rapidly and legitimately[1].

Gwang-Jo Chung et al (2010) states that the most extreme response power for each joint that could be utilized for static inflexibility investigation. And in the mode examination, they assessed the characteristic recurrence for the general gathered structure and contrasted it with the exploratory outcome with distinguish the precision and the unwavering quality of the FEM models [2].

Zhijun Wu et al(2011) fabricate two concurring 3D limited component models of the compartment crane by ANSYS individually dependent on two regular door jamb structures of compound kind and single support type. And afterward stress, twisting, mode shapes are dissected and looked [3].

Jeevan et al(2015) contemplates the demonstrating and investigation of robot arm utilizing ansys . The mechanical plan, basic examination, and results confirmation of another superior semi-direct drive robot arm. A plan enhancement technique utilizing limited component investigation (FEA) is assessed, and a subsequent arm configuration is accounted for [4].

Liao (2010) states they are chip absent at the base of the welding robot. By utilizing ANSYS-10.0, they are find the characteristic frequencies, mode states of base of welding robot and energetic examination of robot. There comes about from demonstrate examination are, the upper edge and tail edge of the base have a greater vibration which are unprotected to be depleted and hurt. They found most extraordinary bounty totality at the seventh common repeat was 39.249 Hz. By utilizing ANSYS - 10.0 programming the results are rapid and solid [5].

J.H.Varma et al. (2015) this undertaking deal with the robot firearm bolster structure for welding motivation behind light vehicle entryway outline. Right off the bat they plan and dissect welding subsystem with their usefulness and applications. They plan a structure of welding weapon bolster which can move to various areas immediately in any event, causing power in tunes of 1.5 occasions of gravity [6].

R. P. Goldberg et al. (2004) they reports the plan, investigation and relative consequences of basic and FEA Design Analysis and Optimization of Robot Pedestal [7].

G.A.Yadav et al. states that most minimal common recurrence of structured arm is 66 Hz in the 90 arrangement, they are discovered that pinnacle tip speed of arm is more prominent than 6m/s and the increasingvelocities are more noteworthy than 7g with greatest position following mistakes under 5% [8].

A. Objective

- To create the design of Bridge riser based on the inputs and FEA for the design.
- To study overall behaviour of the bridge riser according to the Emergency-Stop of the robot.
- To analyse the equivalent stress and total deformation of bridge riser on various constraints.

2 Problem Identification

In the conceptual design, the structural base riser is used for mounting Fanuc M-900iB/280L, which is assigned for material handling (pick & place). Now, the design validation is necessary for the effective working of the robot according to working area and inputs. So, when the design validation is done, it is found out that the structural base riser is actually having enough strength and excessive material utilization. That directly affects the cost of material and cost of fabrication .Due to excessive material usage the structural base weight is 6700 kg.

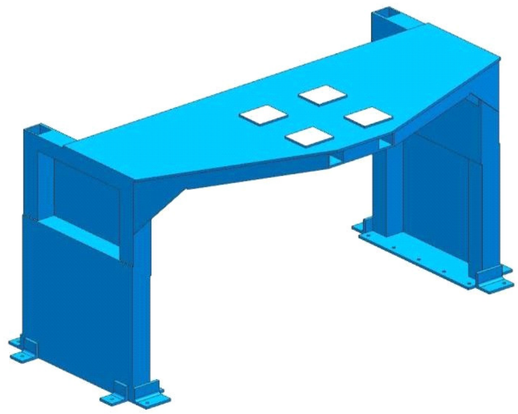

Fig. 1. Existing bridge riser model

TABLE I. MATERIAL PROPERTIES OF BRIDGE RISER

Material	Structural Steel (S235)
Modulus of Elasticity	210GPs
Yield Strength	235 MPa
Poisson’s Ratio	0.30
Density	7850 kg/m3
Riser Weight	6.7 Tonnes

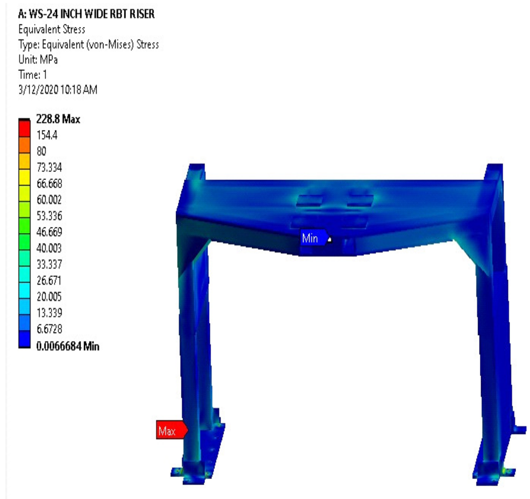

Fig. 2. Existing bridge riser model stress analysis

The Maximum Stress induced in the conceptual model is228.8Mpa

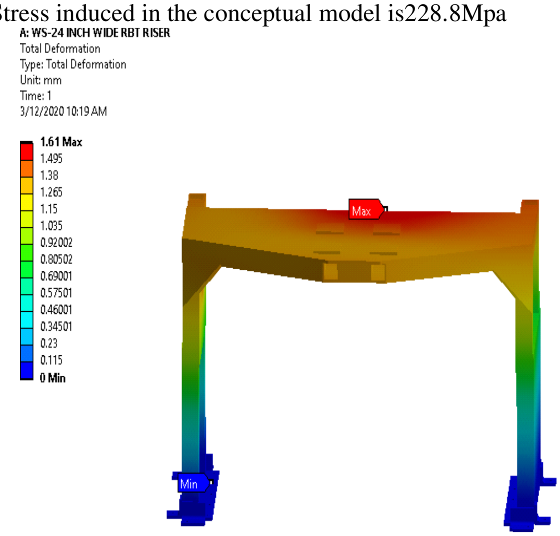

Fig. 3. Existing bridge riser model deformation analysis

The Maximum Deformation occurred in the old conceptual model is 1.61mm

So, main objective of this project is to redesign the model by using the inputs and to optimise the design to have enough strength and reduced material usage.

3 Design And Setup Details

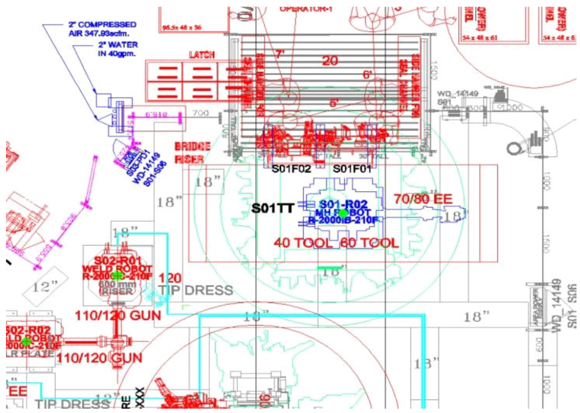

Fig. 4. Location of the bridge riser in layout

Turntable is used in the process. Robot need to be placed above the turn table for better reach. Bridge riser is used to elevate the material handling robot in the layout. Bridge riser avoids collision of robots with turntable. Operator loads the panel on the fixture. Robot picks panel from the load station. Robot rotates 180° and drop panel to next station.

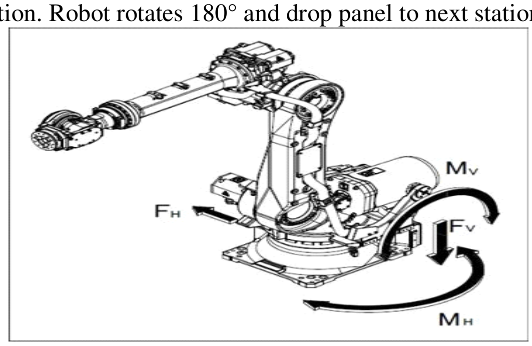

Fig. 5. Various forces on robot

TABLE II.TYPES OF LOAD ACTING ON THE BASE

Vertical moment [MV]	119.56 KNm
Force in vertical direction [MH]	57.82 KN
Horizontal moment [MH]	34.30 KNm
Force in Horizontal direction [FH]	44.10 KN

4 Design Of The Bridge Riser

The bridge riser is to be upgraded from the conceptual design. The design is done by using the NX CAD software

.The bridge riser is to be designed without changing its application. The bridge riser is optimised s by adjusting the sizes of the components in the structure. The unwanted materials in the design were to be removed. Then the tubular structure was used in bridge riser for weight reduction. The turntable is under the bridge riser so the dimensions were done by viewing the constraints from the layout and also by the type of robot to be placed in bridge riser.

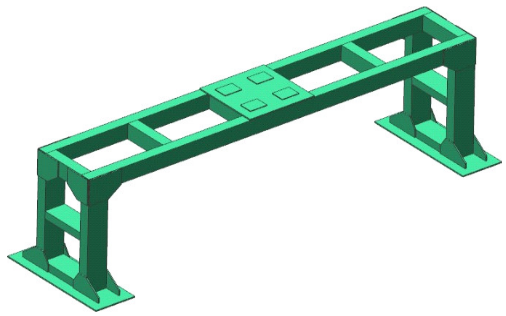

Fig. 6. Bridge riser model

By elevating the robot the reach and range is changed for the robot. Then the bridge riser was set in layout for the feasibility of reach and range.

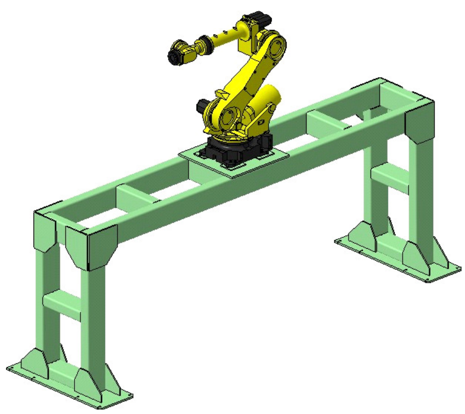

Fig. 7. Bridge riser with robot

The frame weight is optimised by having tubular structure in the design. It has only 3400kg. It has the low weight when compared to conceptual model.

- *Total Force Acting on the Riser*

The total load acting on the bridge riser is Ftotal = Fmax + Fpayload of robot Ftotal = 5886N

Now, considering factor of safety = 2,

∴ Ftotal = 5886 × 2
∴ Ftotal = 11778N
Where Ft- total force acting on riser
Fmax -Maximum load of the robot, Fp – Payload of the robot.

- *Stress Acting on the Riser*

Stress= Ft/A1
Stress = 123*106N/mm
Where Ft- total force acting on riser A1-Area of the bridge riser

The total load [Ft] is calculated by adding the payload of robot and robot load. Then stress is calculated Force by Area. The force is total force which we calculated earlier and then the Area of the bridge riser were calculated and stress were calculated.

5 Results And Discussions

- *Finite Element Analysis of Bridge Riser*

The model is to be analysed by using ANSYS Workbench 2019 R2.The material is selected as structural steel. Then the von misses stress and maximum deformation was to be analysed.

For the static analysis mesh strategy are given in below table.

TABLE III MESH STRATEGIES

Element size	10mm
Nodes	550174
Elements	84547
Mesh type	Hex Dominant

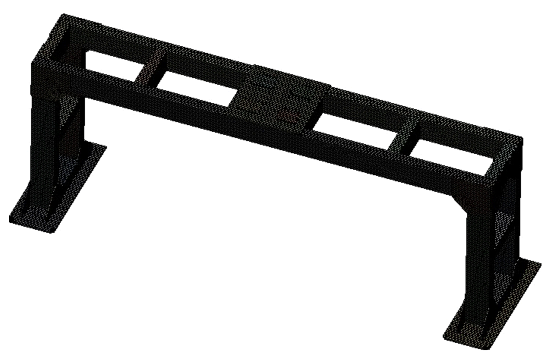

Fig. 8. Meshing in Bridge riser

Then the boundary condition was given to the model as per the E-stop condition of the robot. Thus Estop condition is the worst condition of the robot and the maximum forces and moments induced at this condition. Then supports for the riser were given in this state.

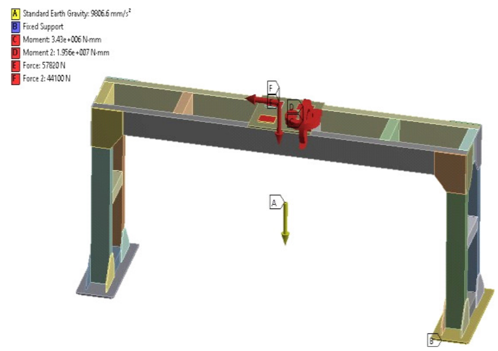

Fig. 9. Boundary conditions for the Bridge riser

Then by applying the boundary conditions the various result plots were selected. The various results to be viewed are Von misses stress and maximum deformation. Then the results are to be solved.

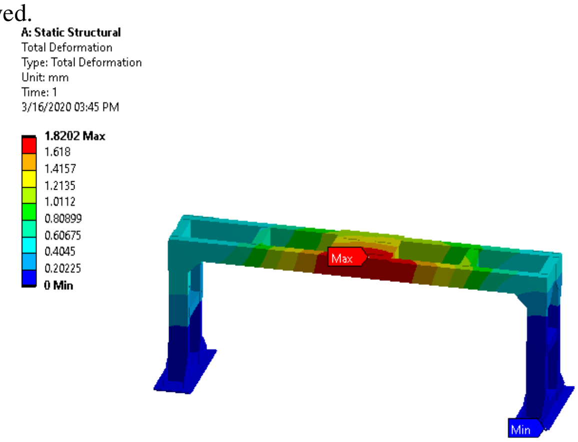

Fig. 10. Total deformation of Bridge riser

The maximum deformation attained in the bridge riser model is 1.8mm.This is less compared to the conceptual model. The Maximum stress induced in middle of the riser at the mountings of the robots .The minimum acts on the grouting area in the bottom leg of the riser.

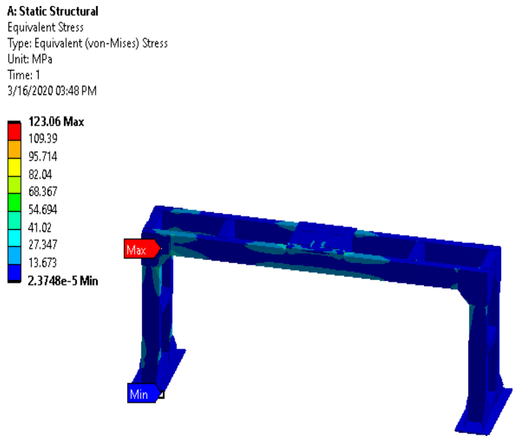

Fig. 11. Equivalent stress in the Bridge riser

The Maximum Deformation occurred in the old conceptual model is 1.61mm

So, main objective of this project is to redesign the model by using the inputs and to optimise the design to have enough strength and reduced material usage.

6 Comparison of results

TABLE IV. RESULT COMPARISON

Results	Equivalent Stress(Mpa)	Total Deformation(Mm)
Existing Model	228	1.61
New Conceptual Model	123.06	1.8

The result comparison shows that the new conceptual model has the low equivalent stress whereas the existing model has 228 MPa and is comparatively high. The newconceptual model has the 2 times Factor of safety. So it is more reliable and it can withstand more load .So these bridge risers were used for different application robots and also at different places.

The total deformation in existing model is 1.61mm and the new conceptual model has the deformation of 1.81mm

.This new conceptual deformation value is slightly high when compared to the exiting model .The allowable deformation is 5mm so the 1.81 deflection doesn’t make any problem to the design.

The optimized structural base frame weight is reduced about 50 compare to existing structural base fame. The New optimized structural base frame weight is about 3400 kg where, the existing structural base frame weight is 6500kg.

The tubular structures were used in the new conceptual model. Due to this reason the optimized structural base frame is lighter than the existing one so, the optimized structural base frame is having reduced material utilization in the fabrication. So, it also gives the cost cutting in the overall cost of the structural base frame.

7 Conclusion

An effort is taken for the Plan and Investigation of the Basic base outline for robot mounting. At first Plan of basic base outline is optimized by utilizing measuring strategy of auxiliary optimization strategy. Unused optimized basic base outline CAD show is created in NX CAD program. Inactive comes about are carried out by Ansys Program and created optimized plan of basic base outline for robot mounting. In unused optimized plan of auxiliary base outline, is concluded that unused optimized auxiliary base outline is in fact advocated and demonstrated its viability over an existing auxiliary base outline.

References

[1] Xiaoping Liao, Changliang Gong –"The Finite Element Modal Analysis of the Base of Welding Robot"; 2010 3rd International Conference on Advanced Computer Theory and Engineering (ICACTE), 2010 IEEE

[2] Gwang-Jo Chung, Doo Hyung Kim; "Structural Analysis of 600Kgf Heavy Duty Handling Robot Principal Research engineer in Robotics Lab". Korea Institute of Machinery & materials Daejon, Korea, 2010 IEEE.

[3] Zhijun Wu, Kailiang Lu and HuiqingQiu, "Finite element comparative analysis of two doorframe structures in container crane," 2011 Second International Conference on Mechanic Automation and Control Engineering, Hohhot, 2011, pp. 302-305. doi: 10.1109/MACE.2011.5986918

[4] Jeevan, dr. amarnageswararao,"modeling and analysis of robot arm using ansys" ijsetrissn 2319-8885 vol.04,issue.33, august-2015, pages:6692-6697

[5] Xiaoping Liao, Changliang Gong " The Finite Element Modal Analysis of the Base of Welding Robot"; 2010 3rd International Conference on Advanced Computer Theory and Engineering (ICACTE), 2010 IEEE.

[6] - Jatin H. Varma, Onkar G. Sonare-" Analysis and Optimization of Robot Gun Support Structure for Welding of Light Weight Vehicle Door Frame", ISSN (Print) : 2321-5747, Volume-3, Issue-6,2015

[7] Roy, R. P. Goldberg and L. L. Whitcomb, "Structural design, analysis, and performance evaluation of a new semi-direct drive robot arm: theory and experiment," in IEEE/ASME Transactions on Mechatronics, vol. 9, no. 1, pp. 10-19, March 2004. doi: 10.1109/TMECH.2004.823848

[8] G.A.Yadav, A.M.Pirjade, M.M.Jadhav,VinaayPatil " Design Analysis and Optimization of Robot Pedestal" International Journal of Engineering and Technical Research (IJETR) ISSN: 2321-0869 (O) 2454-4698 (P), Volume-4, Issue-4, April 2016.

[9] D. S. Vijayan and J. J. Daniel, "An investigation on the torsional effect of symmetric moment resisting frame system subjected to eccentric reinforced concrete lift wall - A finite element approach," Int. J. Eng. Trends Technol., vol. 69, no. 8, pp. 179–184, 2021, doi: 10.14445/22315381/IJETT-V69I8P222