Analysis and Design of General Bridge Crane Structure using CAD Technology

Xinzhong Chang[1] and Amit Sharma[2]

[1]Henan Vocational College of Applied Technology, Henan, Zhengzhou, 450042, China
xinzhongchang@outlook.com
[2]Department of Computer Science and Engineering, Chitkara University, Punjab, India
amit.amitsharma90@gmail.com

Corresponding author: Amit Sharma, amit.amitsharma90@gmail.com

Abstract. In order to reduce the workload of mechanical designers, this paper takes the general bridge crane as an example. Based on the traditional design mode, the parametric technology such as CAD technology, CAE technology and other design methods are tested. The aim of this study is to reduce the complex calculation and heavy workload of the bridge structure; therefore the secondary development is carried out with C programming language. This paper introduces the function of each step and the related functions. Through the case analysis of bridge structure, combined with the traditional structure design theory, the analysis results are compared. The experimental results show that the software realizes the functions of structural design, parametric modeling, parametric automatic assembly and parametric finite element analysis.

Keywords: Bridge crane structure; Caddcae; Parameterization.
DOI: https://doi.org/10.14733/cadaps.2022.S2.15-25

1 INTRODUCTION

Bridge crane is widely used in machinery industry, industrial and mining enterprises and other fields, and plays an important role in national production. It has the advantages of large bearing capacity, high reliability and relatively simple manufacturing process. With the continuous development of cranes to large tonnage, high parameters, low noise, small vibration and lightweight direction, the original finalized products are more and more unable to meet the current requirements. The market needs good performance, good economic bridge crane. The main purpose of bridge crane is to reduce the weight of the whole crane by 60%.

The drawing software of CAD design system used by domestic enterprises is mostly two-dimensional drawing software. There are many shortcomings in the practical application of two-dimensional drawing in design analysis, which cannot be recognized by the new finite element analysis software. However, when using the finite element software to carry out the finite element modeling analysis, it is necessary to reconstruct the three-dimensional model to make the product more visual, which leads to the lengthening of the development cycle. In order to meet the specific needs of the enterprise and achieve its own characteristics, it is necessary to develop a professional CAD design system suitable for the enterprise itself. With the continuous strengthening of design requirements, on the basis of traditional design methods, combined with modern design ideas, parametric technology, CAD technology, CAE technology, the structure of bridge crane is studied. For example, in the design process of the series general bridge design crane, the designers use a lot of time to carry out repetitive work, and the idea of computer-aided design can greatly shorten the design cycle [1].

Using computer to design the shape and decoration of packaging and the structure of cushioning packaging can not only shorten the design period, improve the design efficiency and precision, but also save the input and material resources. The application of CAD technology in the field of packaging industry mainly includes: design the structure of sales packaging container; reasonable selection of packaging materials; prediction of shelf life; calculation of stacking height of goods; reasonable design of storage and transportation stacking arrangement scheme of goods; realization of auxiliary design and scientific evaluation of packaging scheme. In this paper, the C programming language is used to program the proposed method. Net platform in Visual Studio 2012 development environment and the modeling and analysis system of bridge crane skew rail box girder is developed. Solid works 2012 is used as the support software for drawing the model, and the simulation plug-in of solid works 2012 is used to carry out the finite element analysis of the model structure.

The rest of the manuscript is organized as the recent work in the design of general bridge crane using CADDCAM system is described in Section 2. The research methods includes establishment of finite element analysis and the realization of finite analysis is discussed in Section 3. The experimental results and its implementation is described in Section 4. Further in section 5 the conclusion of the proposed bridge crane structure is presented in Section 5.

2 LITERATURE REVIEW

In the past research, the application of two-dimensional parametric CAD technology combined with the design of bridge crane makes the design idea relatively perfect, and solves the problem of repetitive work encountered by designers in the process of bridge crane design. Ling et al. [2] used Auto CAD to study the gantry crane and realized the parametric design and parts design. Mcdougall et al. [3] used full parameter design, and developed a professional CAD design software for general bridge crane by using Auto CAD. The software provides parametric drawing of various series and models of general bridge crane.

Wardhana et al. [4] developed a CAD design system for bridge crane on the platform of Auto CAD two-dimensional design software. On the basis of drawing, structural analysis was added and parametric optimization design was carried out by orthogonal grid method. The combination of optimization analysis and two-dimensional parametric drawing was realized. Antsev et al. [5] applied the modular design and 3D parametric CAD technology to the design and development of gantry crane, and proposed the management method of modular design in the design of gantry crane and the 3D parametric design method for the module of gantry crane. Wang et al. [6] and others studied the three-dimensional parameterization method of bridge crane and gantry crane through the secondary development of solid works 3D drawing software.

Recently, some scholars have studied the dynamics of crusher and the dynamic balance of ring hammer by means of measurement test. However, there are few researches on the motion simulation and finite element analysis and optimization of crusher by modern design optimization methods [7]. The time of analysis, calculation and design can be greatly saved by using the means of motion analysis and finite element analysis, and the motion track, stress situation and vibration state of crusher can be clearly displayed [8, 9]. At the same time, we can make full use of the results of correlation analysis to carry out rapid and efficient structural optimization design of the crusher, in order to improve the comprehensive performance of the crusher, and also provide reference for the improvement and optimization of similar crushers [10, 11].

In this paper, SolidWorks is redeveloped based on C programming language. Through parametric modeling of the bridge structure of general bridge crane, and using simulation software to achieve parametric finite element analysis, the modeling and finite element analysis are integrated on the same platform to achieve the integration of modeling and analysis software. It can reduce the repetitive operation in manual drawing and finite element analysis, and improve the design efficiency.

3 RESEARCH METHODS

Bridge crane design and its calculation is a complex process and it consumes large amount of time of engineers. To overcome this restriction the idea is to design and interactive information based smart system m that it is capable for faster calculations for bridge crane elements, the automatic process of geometric model development and the analysis of stress deformation of bridge crane girder is studied. The architecture of the proposed design for bridge crane structure is depicted in figure 1. The first phase includes the entering of input variables and based on these input data the girder are computed. In the next step the standard girders adopted and the non-standard girders are transferred back to the adoption phase. The next two phases consist of calculation of girders on the basis of stress and deflection. After the calculation of deflection and stress parameters the standard girders are adopted and based on these parameters, calculation report is generated. The final geometrical model is obtained by utilizing the information of the calculation report.

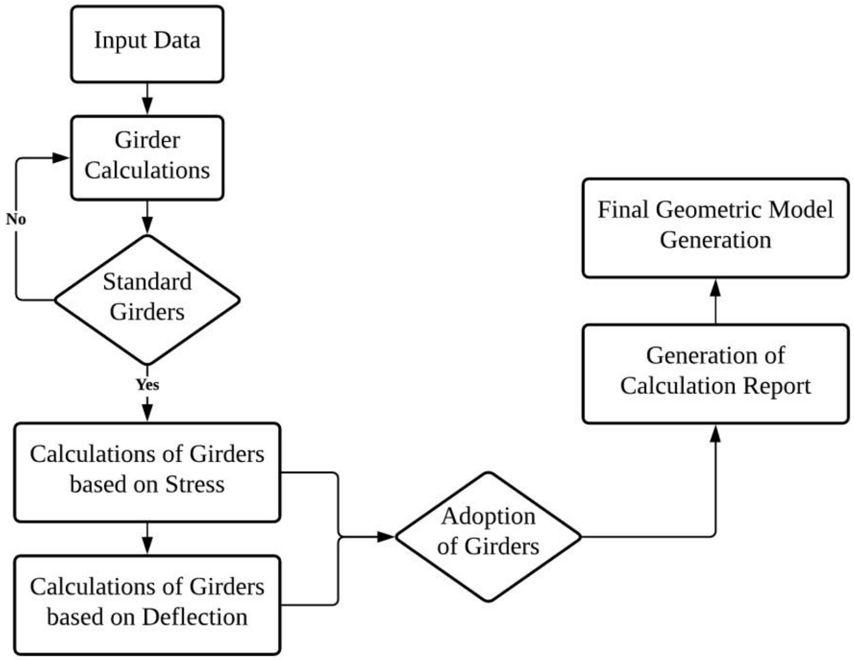

Figure 1: Framework for structure of general bridge crane.

The above discussed issue is addressed by designing the information based smart system for calculating and designing the bridge crane using CAD. In order to evaluate the main girders from bridge cranes the initial step is to check the forces applied on them. The proposed model is parameterized for the realization of its integration to a smart CAD system from its calculations for the generation of final geometrical model.

3.1 Bridge Structure Design of General Bridge Crane

There are many bridge structure types of bridge crane, which can be divided into single beam and double beam according to the number of main beams. According to the bridge structure, it can be divided into three types: steel beam type, box structure type and truss type. The structural modeling of the main beam determines the bridge structure. At present, there are two main types of bridge at home and abroad, which are four truss Type Bridge and box Type Bridge. The box structure has the characteristics of simple manufacturing process, small structure height, convenient maintenance and installation.

According to the layout position of trolley track on the box double beam bridge, it can be divided into three types: the box girder bridge with normal rail is located in the middle of the two webs (Fig. 2a), the one on the main web is the skew rail box girder bridge (Fig. 2b), and the semi skew rail box girder bridge (Fig. 2c) is arranged between the web and the center line of the main beam.

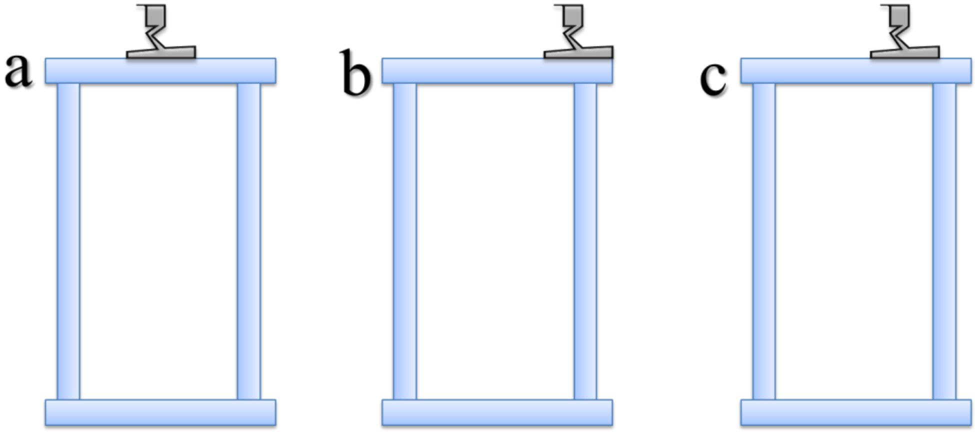

Figure 2: Box bridge structure (a) Middle rail box girder bridge (b) Skew rail box girder bridge (c) Semi skew rail box girder bridge.

In contrast, the skew rail box girder bridge has the advantages of better vertical and horizontal rigidity and smaller bending deflection. Moreover, under the same service conditions, the structure is lighter and the bearing capacity is stronger, so it is widely used in industrial production. In the design of metal structure of Crane Bridge, choosing reasonable design method to get reliable structure is the purpose of any design method. With the continuous in-depth study of scientific theory and the rapid development of computer technology, there are many design methods, such as optimization design, simulation, finite element design, and dynamic simulation design and so on. In this paper, the allowable stress method is used to check the accuracy of the calculation results of the bridge structure.

The allowable stress method refers to the design method of stress and deformation under the condition that the external load does not exceed the bearing capacity (strength, deformation control value and stability resistance) of the structure and connection joint. The design steps can be divided into four steps:

Step 1: Calculate and analyze the load fi of crane and increase it with appropriate dynamic load coefficient Φ I;

Step 2: It is used to get the combined load FJ through the load combination table. Then the internal force is calculated by FJ to determine the resultant load effect sk;

Step 3: According to the load effect on the component or component and combining with the principle of material mechanics, the stress σ L1 is calculated and combined with the stress σ 2L caused by local effect to obtain the composite design stress σ L;

Step 4: Compare the resultant design stress σ l with ADM σ. When σ l < ADM σ, the calculation is qualified, otherwise it is unqualified. The allowable stress ADM σ is determined by removing the strength R from the strength coefficient γ F. in some cases with high risk, it needs to be divided by the high-risk coefficient γ n.

When the allowable stress method is used in structural design, the relationship between external load and internal force is usually linear. The external load and internal force of bridge crane are linear [12].

3.2 Establishment of Finite Element Model and Realization of Finite Element Analysis

The geometric model of finite element analysis using simulation plug-in module in solid works comes from the parts and assemblies drawn by Solid Works software. It is required that the geometric model can produce the correct finite element and mesh moderately small. The concept of "moderately small" refers to the number of elements in the grid rather than its actual size. In general, three methods of feature hiding, idealization and elimination are used to modify CAD geometric model to meet the requirements of meshing.

- **Feature hiding:** merge or eliminate the features that are considered unimportant in the analysis, such as fillet, circle edge, logo, etc.
- **Idealization:** idealization is a more positive work, which may deviate from the original shape of a 3D geometric model, such as replacing a thin-walled model with a plane.
- **Clear:** as the name implies, in CAD geometry model, some Multi-Solid or slender faces cause difficulties in meshing, and some even can't mesh, so they need to be cleared.

The structure of bridge crane is modified and simplified as follows:

- All the thin plate structures in the bridge crane model are idealized as plane elements, that is, shell elements;
- The wheel on the end beam is simplified as support point constraint;
- The trolley mechanism of the crane is simplified as a concentrated node force acting on the main beam rail;
- The idealized loading method is used to solve the problem. Use the solver of solid works simulation to get some desired result data.

Feature driven modeling is equivalent to the manual operation of solid works modeling steps through computer programs, and the three-dimensional modeling process is completely controlled by the program. Modeling has strong flexibility and does not need model base as support. Usually, this method requires a lot of programming and requires developers to have a good grasp and application ability of solid works API, which is suitable for modeling with model size directly related to each other. The simple way to understand the API function is to record the operation process of the model through the macro of solid works, and then wait for the corresponding usage of the required function through the API. This method is helpful for understanding parametric modeling and using solid works API functions. The related code library of CSLD is as follows:

```
public void Extrusion(double height)
{
    bool boolstatus = false;
    Feature my Feature = null;
    my Feature = ((Feature)(sw Doc.Feature Manager. Feature
    Extrusion2 (true, false, false, 0, 0, height, 0, false, false,
    false, false, 0, 0, false, false, false, false, true, true, true, 0,
    0, false)));
    boolstatus = sw Doc.Extension.Select By ID2("Point1@
    origin", "EXTSKETCHPOINT", 0, 0, 0, false, 0, null, 0);
    sw Doc.Blank Sketch();
    sw Doc.Clear Selection2(true);
}
```

Through parametric 3D modeling technology, the 3D model of general bridge Crane Bridge is imported into the finite element plug-in simulation of solid works for finite element analysis, and a static calculation example is created. The code for importing the bridge crane model into solid works simulation model is as follows:

```
string Path = Directory.Get Current Directory();
SW doc = ((model DOC2) (SW app. Open doc6 (path
+ @ "- Parts Library \" + "bridge crane. Sldprt", 1,
0,"", ref longstatus, ref longwarnings)));
```

Part code of creating static calculation example:

```
CWObject = (Cw Addincallback)sw App.Get Add
In Object("Sld Works.Simulation");
COSMOSWORKS = (Cosmos
Works)CWObject.Cosmos Works;
Study = (cwstudy) study mngr. Create new
Study2 ("bridge crane analysis"),
(int)sws Analysis Study Type_e.sws Analysis
Study Type Static, out err Code);
```

3.3 Software System Structure

The software structure of this paper is a design system composed of bridge structure design, bridge overall modeling, bridge assembly, finite element analysis and design specification. In this paper, the visual studio is used as the development platform of Microsoft.

The integration technology in the process of software development is fully used, including the secondary development technology of solid works 3D design software. The word document in office calls and modifies it. The design system is divided by the idea of dividing modules. The overall operation structure of the software is shown in Figure 3 [13].

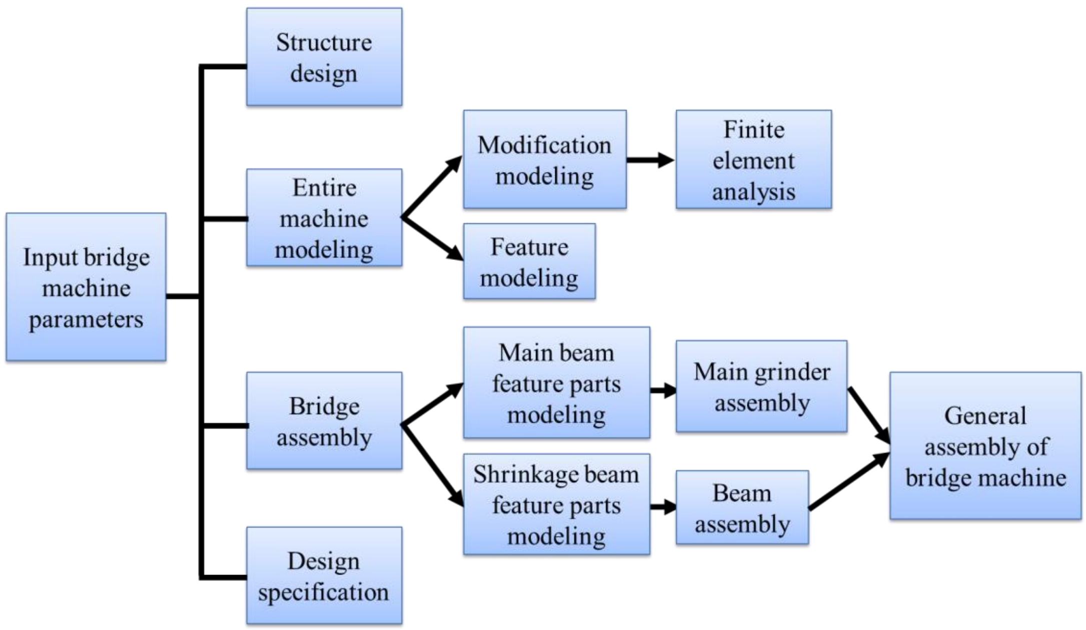

Figure 3: Overall operation structure of software.

Each module is independent and interrelated with each other. Independence means that a single module can output parameters independently and operate independently. Connection means that they are integrated into one, which can determine the bridge structure size of general bridge crane from parametric structural design. Then generate a single independent parametric component for assembly or generate a bridge overall model, and import it into solid works simulation for parametric finite element analysis. Users can use the software system in whole or in part according to actual needs.

4 EXPERIMENTAL RESULTS AND SYSTEM IMPLEMENTATION

4.1 Comparison of Finite Element Analysis Results

The parametric design and development of solid works simulation is carried out, and the parametric modeling of bridge crane bridge structure is carried out according to the corresponding actual engineering data.

The bridge model is parameterized according to the operation steps of simulation. The material selection is ordinary carbon steel, the constraint is unchanged, the applied load is 228800n, and the grid is divided into mixed grids. The grid is a standard grid with four Jacobian points. The grid cell size is 356.002mm, the tolerance is 17.8001 mm, the total number of nodes is 13703, and the total number of cells is 6191. According to the trolley running in extreme conditions, the main beam structure is analyzed.

Condition 1: The trolley is located in the mid span of the main girder, the trolley runs in the mid span position of the main beam, the bridge is stationary, and the main beam bears the stiffness and strength of the trolley weight and the maximum lifting weight. The results show that the maximum stress of the main beam is 122.3mpa, which is less than the allowable stress of 175Mpa.The deformation is 1 mm, and the displacement meets the requirements. Through the analysis of the bridge structure results, it can be concluded that the bridge structure design meets the design requirements.

Condition 2: The trolley is located at the span end. When the trolley works at the cross end of the main beam, the maximum stress of the main beam is 90 MPa and the maximum displacement is 15.7 mm, which meet the design requirements.

Calculation conditions		*Static strength Maximum stress (MPA)*	*Static stiffness (maximum displacement mm)*
Finite element software Calculation results	Condition 1	122.4	22.2
	Condition 2	123.5	22.9
Traditional mechanical calculation		134.6	23.8
Permitted results		176	31.9

Table 1: Calculation results of bridge structure.

The calculation results of the bridge structure are shown in Table 1. Through the analysis of the results under extreme conditions and the traditional mechanical calculation, and comparing with the allowable results, the difference between the calculation results of finite element software and that of traditional mechanics is 9.07%. The software can be used for modeling and finite element analysis of engineering examples [14].

4.2 Software System Implementation

4.2.1 Bridge structure design

The traditional mechanical checking method is used to design the bridge structure. The results of stiffness and strength of bridge structure can be checked by inputting parameters.

4.2.2 Overall bridge modeling

In this paper, solid works is redeveloped by parametric 3D modeling technology. Click the "bridge modeling" button on the main interface to enter the parametric modeling interface of bridge crane bridge structure. After inputting the corresponding parameters, click the "build model" button to build the three-dimensional model of the whole bridge. Input the address of the bridge structure to be saved in the storage address box of the overall bridge model. The establishment of the model is connected to the finite element analysis to provide the model.

Because the establishment of the bridge model is established by the feature method, it will waste a lot of time in modeling after its modification. According to the actual needs, the size modification method is applied to modify the bridge structure by modifying the corresponding size.

4.2.3 Bridge assembly

By clicking the "bridge assembly" button, the bridge structure can be parametric assembled. First, by clicking the "main beam part modeling" and "end beam part modeling" buttons, the parametric modeling of the main beam structure and end beam structure components is carried out. Then click the "main beam assembly" button and "end beam assembly" button to get the main beam structure assembly and end beam structure assembly respectively. The assembly of the final bridge structure can be obtained by pressing the "General Assembly button of bridge crane", and the saved position can be set in the position frame of parts and assembly body. The assembly of bridge structure is shown in Figure 4.

4.2.4 Finite element analysis

In this paper, through the secondary development of simulation in solid works, parametric finite element analysis of bridge structure is carried out. The parametric 3D model of the whole bridge is imported into the simulation, and the parametric finite element analysis is carried out. The advantage of the whole machine modeling is to avoid the interference in the assembly. Because the bridge structure is analyzed as a shell element, it is necessary to parameterize the thickness design of the shell element. The static strength and stiffness of the bridge structure are analyzed by parametric control of the load and the thickness of each plate of the bridge [15].

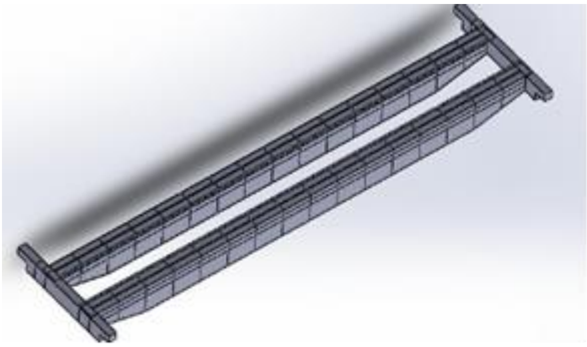

Figure 4: Short, centered caption, terminated with a full stop.

It is observed from the analysis that the overall nonlinear displacement elements framework presents that the maximum displacement is always less than the normal and the second observation is that it makes element out of interest for the consideration of viewpoint of carrier.
The experimentation is carried out to calculate the strength of the Steel structure of Crane Bridge. The model is designed for individual parts of which crane which are further assembled to a single structure. The standard finite element for the creation of each part individually and the complete structure of bridge by utilizing road and plate elements. The observed calculation model for the structure of bridge crane is represented in the figures 5 and 6 below.

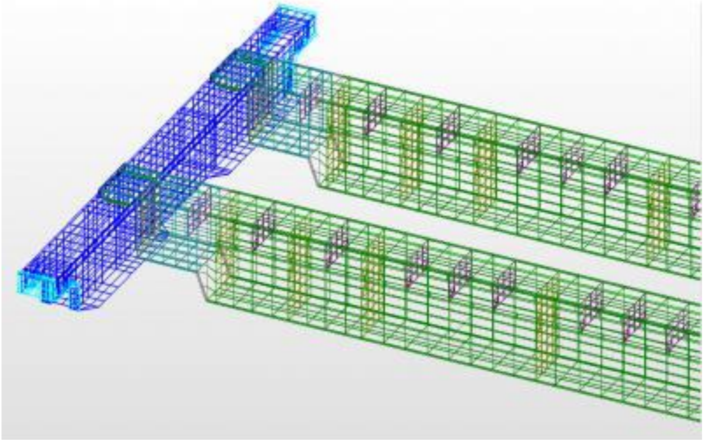

Figure 5: Bridge crane model along with the end span of girders including the internal structures.

Figure 5 represents the complete detail of the designed model along with the end span of girders including the internal structures. Figure 6 represents the displacement of map at vertical axis when the trolley is set to be in middle position of beams of the crane bridge. It is observed from the experimentation that the maximum of stress occurs at beams span in the case when the trolley is said to be in the middle position. Therefore, it is resulted that the construction of Beams requires better strength in the middle section.

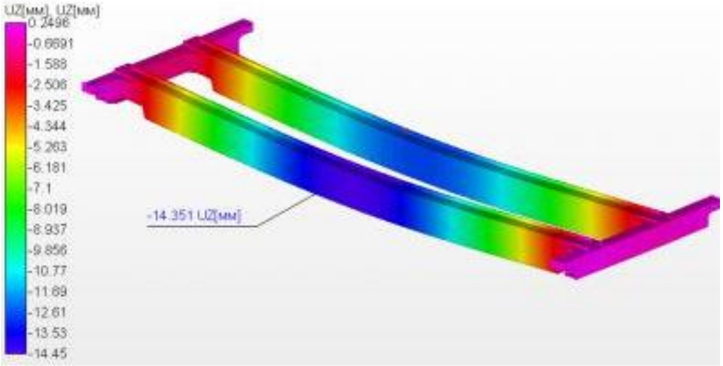

Figure 6: Bridge crane model when the trolley is at middle position.

The level of stress in a model towards wide span and bridge endpoints is very less. It signifies that there is scope of improvement in the design and the metal content can be further reduced. The present work has shown its applicability over the analysis module of structural to find the solution of complex issues of structural analysis of the materials that handles the equipment.

5 CONCLUSION

This paper takes the bridge structure of general bridge crane as the research object. Modular and parametric design method is used to design the bridge structure of bridge crane. The project uses object-oriented technology and C programming language in the development environment of visual studio 2012 for program design. Through the secondary development of Solid Works software function, the three-dimensional model of bridge crane bridge structure is established, and the static strength and static stiffness of dangerous section of bridge girder are analyzed by calling simulation plug-in.

The maximum stress of the girder is less than 175Mpa.The displacement deformation is 30.1 mm, which meets the requirements of stiffness. Through the analysis of the bridge structure results, it can be concluded that the bridge structure design meets the design requirements. When the trolley works at the cross end of the main beam, the maximum stress of the main beam is 90 MPa and the maximum displacement is 15.7 mm, which meet the design requirements. The difference between the results obtained by the finite element method and that of the traditional method is 9.07%. The software can be used for modeling and finite element analysis of engineering examples.

Through the design of the software system, the software can complete the bridge structure design, bridge overall modeling, bridge assembly, finite element analysis and other functions. The designed system can integrate modeling and finite element analysis on the same platform to achieve the integration of modeling and analysis software. It can reduce the repetitive operation in manual drawing and finite element analysis, and improve the design efficiency.

Xinzhong Chang, https://orcid.org/0000-0003-4573-7607
Amit Sharma, https://orcid.org/0000-0002-1700-8848

REFERENCES

[1] Chen, Z.; Li, Z.; Huang, C.; Zhang, G.; Yu, H.: Safety assessment method of bridge crane based on cluster analysis and neural network, Procedia computer science, 131, 2018, 477-484. https://doi.org/10.1016/j.procs.2018.04.235

[2] Ling, Z.; Wang, M.; Xia, J.; Wang, S.; Guo, X.: Stress Analysis for the Critical Metal Structure of Bridge Crane, In IOP Conference Series: Earth and Environmental Science, 108 (2), 2018, 022056. IOP Publishing. https://doi.org/10.1088/1755-1315/108/2/022056

[3] McDougall, J.; Rewerts, L.; Iwand, H.; Stevenson, M.; Peterson, F.; Bell, T.; Karlins, S.: Investigation of a Bridge Crane Derailment with Limited Subject Parts, Journal of Failure Analysis and Prevention, 20(1), 2020, 51-57. https://doi.org/10.1007/s11668-019-00798-5

[4] Wardhana, W.; Hadiwidodo, Y.-S., Siswanto, F.: Strength Analysis and Sea-Fastening Design of Container Crane Structure for Heavy Lifting Sea Transportation, In Applied Mechanics and Materials, Trans Tech Publications Ltd., 874, 2018, 147-154. https://doi.org/10.4028/www.scientific.net/AMM.874.147

[5] Antsev, V.-Y.; Tolokonnikov, A.-S.; Gorynin, A.-D.; & Reutov, A.-A.: A statistical model of operational impacts on the framework of the bridge crane, In IOP Conference Series: Materials Science and Engineering, IOP Publishing, 177 (1), 2017 012053. https://doi.org/10.1088/1757-899X/177/1/012053

[6] Wang, T.; Tan, N.; Zhou, C.; Zhang, C.; Zhi, Y.: A novel anti-swing positioning controller for two dimensional bridge crane via dynamic sliding mode variable structure, Procedia Computer Science, 131, 2018, 626-632. https://doi.org/10.1016/j.procs.2018.04.305

[7] Patalano, S.; Vitolo, F.; Lanzotti, A.: A Graph-based Software Tool for the CAD Modeling of Mechanical Assemblies, In *GRAPP/IVAPP* , 2013, 60-69. https://doi.org/10.5220/0004299000600069

[8] Camba, J.-D.; Contero, M.; Company, P.: Parametric CAD modeling: An analysis of strategies for design reusability, Computer-Aided Design, 74, 2016, 18-31. https://doi.org/10.1016/j.cad.2016.01.003

[9] Chang, K.-H.: Product design modeling using CAD/CAE: the computer aided engineering design series, Academic Press, 2014. https://doi.org/10.1016/B978-0-12-398513-2.00003-8

[10] Sakao, T.; Shimomura, Y.; Sundin, E.; Comstock, M.: Modeling design objects in CAD system for service/product engineering, Computer-Aided Design, 41(3), 2009, 197-213. https://doi.org/10.1016/j.cad.2008.06.006

[11] Kim, J.; Pratt, M.-J.; Iyer, R.-G.; Sriram, R.-D.: Standardized data exchange of CAD models with design intent, Computer-Aided Design, 40(7), 2008, 760-777. https://doi.org/10.1016/j.cad.2007.06.014

[12] Hu, Y.; Wu, B.; Vaughan, J.; Singhose, W.: Oscillation suppressing for an energy efficient bridge crane using input shaping, In 2013 9th Asian Control Conference (ASCC), IEEE, 2013, 1-5. https://doi.org/10.1109/ASCC.2013.6606196

[13] Han, Z.; Tong, Y.; Luan, J.; Xiangdong, L.: Reliability modeling of double beam bridge crane, IOP Conference Series Materials ence and Engineering, 362(1), 2018, 012028. https://doi.org/10.1088/1757-899X/362/1/012028

[14] Wei, Y.; Yifei, T.; Dongbo, L.; Xiangdong, L.; Xiayi, Q.: Design and development of Web-based energy-saving evaluation system for bridge crane, Hoisting and Conveying Machinery, 2013, 06. https://doi.org/10.1155/2013/763545

[15] Sorensen, K.-L.; Singhose, W.; Dickerson, S.: A controller enabling precise positioning and sway reduction in bridge and gantry cranes, Control Engineering Practice, 15(7), 2007, 825-837. https://doi.org/10.1016/j.conengprac.2006.03.005