Efficient method for predicting fatigue of four-linked structural based submodel

Xueling Li¹, Xiaoqiang Guo¹, Hongfeng Ma¹, Genwen Li²
¹Jiangsu XCMG Construction Machinery Research Institute Ltd, 221004 Xuzhou, China
²XCMG. Xuzhou Truck-Mounted Crane Co., Ltd, 221004 Xuzhou, China
E-mail: lixueling861126@163.com

Abstract. An efficient method is proposed in this paper to improve the minimum fatigue safety factor of the four-link system using shape optimization. In the meanwhile, the probabilistic response of the structure on the minimum fatigue safety factor is studied. This paper takes the four-link system as the object of study. The time-consuming simulation model were often employed in conventional method for such complex system that including a various of parameters and decision variables, and this would increase a mass of time cost. In the proposed method, submodel techniques are used to replace the original uneconomical simulation model and reduce the computational burden in complex system performance analysis and shape optimization. In order to certify the accuracy of the submodel, the result of shape optimization based such method was tested by replacing it back into the original HYPERMESH models and recalculate in OPTISTRUCT. The result of which indicated that it was attractive in finding out the optimum shape for improving minimum fatigue safety factor. Probabilistic analysis was employed for the purpose as well with vast experiments. In this paper, utilizing submodel technique can save approximately two-thirds of the time, and the result of which is practically the same with the original model.

1. Introduction

The four-link system is very important in engineering plant especially for excavator. The performance of four-link system can directly determine the capability of the excavator. Substantive experiments indicate that four-link system is prone to fatigue failure. A large number of studies show that fatigue is a localized structural damage phenomenon and occurs when the material is subjected to cyclic loading. Fatigue fracture is the main failure mode. In order to improve the fatigue life of four-link system, we need to design components with durability at the process of design stage. Increasing the minimum fatigue safety factor (FSF) is an effective method to guarantee the fatigue life and to achieve such goals; the applications of CAE based shape optimization to fatigue are employed in this paper. Such as the commercial software, HYPERMESH and OPTISTRUCT, are used to build the finite element (FE), and perform the shape optimization respectively.

There are many studies in literature where shape optimization and finite element analysis (FEA) were successfully utilized to improve the fatigue life of the structure. Miche et al. [1] presented a method based local fatigue failure of continuum structures in topology optimization to redesign a fail-safe structure which remained operable in a damaged state. They adopted a simplified model for local fatigue failure and the minimax formulation of the topology optimization was utilized to minimize the worst case performance. Ibrahim and Kharmanda [2] presented a new methodology of reliability-
based design optimization (RBDO) which leading to a global solution without adding additional computation time. The safety factor formulation for nonlinear problem was used to improve the computational efficiency. In order to indicate analytically the efficiency of this methodology, the optimality conditions are used by them. They had successfully certified the advantages of dealing complex structural problems by using the optimum safety factor (OSF). Peng [3] proposed an innovative approach based shape optimization of three-dimensional, damage tolerant structures. In this strategy, a simple and new method, which termed Failure Analysis of Structures (FAST), was used to evaluate the stress-intensity factor for cracks at a notch. Through using FAST, the worst crack locations can be obtained by modeling cracks along cracked critical edges of the structure [4]. The computation time is one of the most challenge when perform the shape optimization for finite element (FE) with complex structure and huge amount of units. In order to decrease the computation time and sustain even increase the accuracy of the result, a submodeling or substructuring technique was introduced in this paper. With submodeling technique used, the number of elements in the finite element model (FEM) can reduce rapidly as the part of the FEM which is not used for inputting the boundary and outputting the computed result are replaced by the substructures. Submodel method is used by Xu [5] to check the bolt strength of marine equipment and the results of the calculations based this method accorded well with the conventional means. Ahsan [6] successfully characterized an application of a piezoresistive sensor based MEMS pressure sensing under variations of both orientation and size by using submodel technique. He proposed three advantages of using this method, efficiency, economizing computation time and simplicity of post-processing. Liu [7], had improved the global fidelity of submodels for multi-level system design optimization and performance analysis by using a new sequential sampling strategy.

![Fig. 1 The experiments of four-link system:](image)

a The full range of conditions, b The critical condition c The local figure of the experiment
In order to obtain the precise design result, it is necessary to consider the structure of the four-link system and the transfer path of loading. The four-link system is used to connect the bucket to the excavator arm and transmit the hydro-cylinder force.

In this paper, the commercial software, HYPERWORKS, is introduced to deal the simulation problem. The purpose of this paper is to progress an efficient strategy for the fatigue based shape optimization of FEM including substantive number of degrees of freedoms and nonlinearities, and to calculate the probabilistic response of the minimum FSF for the four-link system considering the variation of the hydro cylinder forces.

2. Full FEM and Submodel

2.1 Experiments and Full FEM

The experiment in full range of conditions is shown in Fig. 1, it can be seen from the Fig. 1 (a) that the two power cylinders are installed on the left hand side of the four-link system and on the opposite side, a crossbeam is used to replace the initial hydro cylinder. The direction and magnitude of composite hydro cylinder force can be adjusted by changing the angle and magnitude of the two power cylinders. The critical angle of the crossbeam and power cylinders can be obtained through experiments and the length of expansion amount for the hydro cylinder is utilized to distinguish the critical working condition. The special experiment for the critical working condition is performed as shown in Fig. 1(b). After 221k times of loading cycles, the fatigue fracture occurred on the connecting rod. Therefore, high cycle stress fatigue analysis can be utilized to predict the fatigue life of the four-link system [8, 9].

![Fig. 2 The finite elements model.](image1)

![Fig. 3 The process of modeling submodel of objective.](image2)

The original model used for fatigue and stress analysis are composed of power cylinders, ear plates, support plates and connecting rod, bearing and bearing sleeve as shown in Fig. 2. The hexahedron elements are employed to build the FEM and there are approximately 295356 elements and 147638 nodes constitute the initial model. As shown in the Fig. 2(b), the FEM was fixed for the translations in x, y, and z directions at support plates installed on the both sides of the connecting rod and released its rotational degrees of freedom and on the hydro cylinder using rigid link elements. 220kN hydro cylinder force, which is the magnitude of the load under the critical working conditions, was applied for the analysis. The length of the expansion amount for hydro cylinder is variable and the scope of the length is 456 mm, which is divided into 4 parts according to the practical working condition. Many nonlinear contact interactions and the variation of direction and magnitude of the hydro cylinder force lead to an increase in the burden of computation. One OPTISTRUCT solution often lasts approximately 20h when 8 CPU’s were used in parallel calculation.

The connecting rod, as shown in Fig. 3c, was modeled by second order hexahedron elements due to the application of second order element can get a more precise result by eliminating unnecessary approximations of the nonlinearity in the following processing [10].
2.2 Submodel
After the input parameters of the four-link system determined, a submodel was employed in modeling in the local region. There are many obvious advantages in using submodeling technique. First is efficiency as the analysis can be performed directly with a given force without having to resolve the composite force solution due to the boundary conditions have been written to an output text file and a HYPERMESH script was written to read it. The second advantage is that the limited number of elements can tremendously reduce the computational burden. The last one is that the post-processing is simplified as only submodeled elements need to be deal with.

3. The analysis of submodel
The results of fatigue analysis showed that the extreme loading condition occurred when the length of the expansion amount for hydro cylinder is 278 mm and the minimum FSF was found to be 0.922 as shown in Fig. 4. Considering the lowest tolerable FSF is 1.15 for the four-link system, the design for improving the FSF is needed. As mentioned above, since the application of conventional method would take too much computation time, then a more efficient method was employed in this paper based shape optimization using the submodel in a connecting rod of the four-link system to decrease the computational time and effort.

Fig. 4 The min FSF calculated by HYPERMESH on the four-link system for the original model
The submodel, used for fatigue analysis based shape optimization in HYPERWORKS, was built after determining the location with the lowest FSF (the connecting rod). The following method was utilized to found the submodel. For the four load steps where the maximum force was applied with the expansion amount of 278 mm, the relative position of nodes at the boundary of the submodel were recorded to an output text file. The magnitude of the maximum stress and location of the critical region calculated by the submodel were found to be almost the same to those calculated by the original FEM when the second order hexahedron elements were utilized in the submodel.

Table 1. The coefficient of each shape calculated during the process of fatigue analysis

| Iteration | Shape 1 | Shape 2 | Shape 3 | Shape 4 |
|-----------|---------|---------|---------|---------|
| 0         | 0.0000  | 0.0000  | 0.0000  | 0.0000  |
| 1         | 0.1000  | 0.1000  | 0.1000  | 0.1000  |
| 2         | 0.2000  | 0.2000  | 0.2000  | 0.2000  |
| 3         | 0.3000  | 0.3000  | 0.3000  | 0.3000  |
| 4         | 0.4679  | 0.4682  | 0.4681  | 0.4599  |
| 5         | 0.4952  | 0.5109  | 0.5337  | 0.4601  |
| 6         | 0.4983  | 0.5719  | 0.6509  | 0.4721  |
| 7         | 0.5001  | 0.5722  | 0.6510  | 0.4725  |

After finishing the preparation of the submodel, HYPERMESH was employed to morph the FEM of the submodel and these deformed shapes will be utilized as design variables in the process of shape optimization for fatigue. Contrast the reduction of computational time with the quality of the FE
morphing, second order hexahedron elements were replaced to first order hexahedron elements, and the latter were used in fatigue analysis based shape optimization in HYPERWORKS.

The calculated optimal shapes are shown in Fig. 5. The multipliers of variables during fatigue analysis based shape optimization using submodel are proposed in Table 1. The optimal shape of the connecting rod is expected as a combination of four shapes as Fig. 5. These different layouts are the optimal shape of the four conditions. An acceptable layout for the full range of working conditions could be obtained with more optimization iterations and computation time utilizing traditional method while only 9 optimization iterations are needed by using the method mentioned in this paper.

After obtaining the optimal liner combination of these four shapes, the attractive layout of connecting rod using submodel for fatigue based shape optimization was replaced back to the initial finite element model. Analysis was carried out for the optimal shape. The minimum FSF increased from 0.922 to 1.241 for the optimal shape as shown in Fig. 6 for the critical working condition.

The upper and lower bounds of the load applied on the four-link system are 220kN and 70kN respectively in the process of fatigue analysis based shape optimization. In order to guarantee the accuracy of FEA, the influence of the variation on the fatigue life of the four-link system should be considered, and probabilistic analysis was employed to perform the fatigue. The design procedure works as follows: first, the design of experiment was utilized to prepare the 16 runs with the 4 working conditions, and the stress values of each working conditions on the joint region between connecting rod and bearing were shown in Table 2. Fatigue and stress analysis for the four conditions were performed by HYPERMESH for the engine assembly FEM with both optimized and original shapes. The predicted minimum FSF for the critical angle are listed in Table 3.

4. Reliability analysis
The upper and lower bounds of the load applied on the four-link system are 220kN and 70kN respectively in the process of fatigue analysis based shape optimization. In order to guarantee the accuracy of FEA, the influence of the variation on the fatigue life of the four-link system should be considered, and probabilistic analysis was employed to perform the fatigue. The design procedure works as follows: first, the design of experiment was utilized to prepare the 16 runs with the 4 working conditions, and the stress values of each working conditions on the joint region between connecting rod and bearing were shown in Table 2. Fatigue and stress analysis for the four conditions were performed by HYPERMESH for the engine assembly FEM with both optimized and original shapes. The predicted minimum FSF for the critical angle are listed in Table 3.
Table 2. The stress of critical region under the four working condition.

| Critical region | shape 1(Mpa) | shape 2(Mpa) | shape 3(Mpa) | shape 4(Mpa) |
|----------------|-------------|-------------|-------------|-------------|
| Condition 1    | 212         | 177         | 178         | 165         |
| Condition 2    | 424         | 353         | 361         | 303         |
| Condition 3    | 371         | 348         | 349         | 299         |
| Condition 4    | 391         | 324         | 311         | 268         |

Table 3. The min FSF of original shape and optimized shape

| Run | Min FSF of Original shape | Min FSF of optimized shape |
|-----|---------------------------|---------------------------|
| 1   | 1.09                      | 2.12                      |
| 2   | 0.82                      | 1.16                      |
| 3   | 0.85                      | 1.16                      |
| 4   | 0.93                      | 1.31                      |

In order to evaluate the validity of the models, the theory of cross validation was employed in this study. Coefficient of determination ($R^2$) was calculated as following formula:

$$SS_{tot} = \sum_i (\bar{y} - y_i)^2$$

$$SS_{err} = \sum_i (y_i - f_i)^2$$

$$R^2 = 1 - \frac{SS_{err}}{SS_{tot}}$$

Where $\bar{y}$ represents the mean of the min FSF values calculated by FEA, $y_i$ are the 16 min FSF values obtained by FEA, and $f_i$ represent the min FSF values calculated by separate models that are built by satisfying all data points except the $y_i$. The $R^2$ of the original and optimal models were calculated respectively as 0.96 and 0.91. The Monte Carlo simulation was carried out to calculate the statistical distribution of the minimum FSF at the critical angle of load for both the original and optimal shapes by these models with response surface. Normal distribution was employed to simulate the variation of the four working conditions with an appropriate value of 338 kN and 5 kN as a deviation, where the 5kN-deviation is the value away from the mean value of the lower and upper bounds for the four working conditions. The minimum allowable FSF is 1.15. The probability of obtaining minimum FSF lower than 1.15 was 0 and 1 for the optimal and original geometries respectively.

5. Conclusions

A new four-link system with an improved connecting rod was designed by applying fatigue based shape optimization utilizing submodeling technique. Through shape optimization, a new and similar layout of connecting rod was obtained as an optimal design to have improved minimum FSF. By applying the submodeling technique proposed in this paper, the local optimization problem of complex model was solved and as a complicated model with large numbers of elements and nodes was replaced by a simple model, the computational time was tremendously reduced.

The analysis results of OPTISTRUCT based first order hexahedron elements, second order and originals were compared, it was found that the critical region remained the same in both analysis. Hence, the first order hexahedron was used in shape optimization in HYPERMESH as it was more efficient.

The slight variations of the location shape are usually neglected when analysing the complicated structure. However, such variations often have a significant influence on the stress transmitted through the connecting rod surface and on the minimum FSF for the four-link system. It was found to be obviously against the variation of the stress on the surface of connecting rod for the design with the optimal layout.
References

[1] Jansen M, Lombaert G, Schevenels M, Sigmund O. (2014). Topology optimization of fail-safe structures using a simplified local damage model, Structural and Multidisciplinary Optimization 49: 657–666.

[2] Mohamed Haisam I, Mohamed Ghas K, Abderafi C. (2015). Reliability-based design optimization for fatigue damage analysis, International Journal of Advanced Manufacturing Technology 76: 1021–1030.

[3] Peng D, Jones R. (2009). A CAD-based on biological method for designing optimal fatigue life, Structural and Multidisciplinary Optimization, 37: 295–304.

[4] Yasiny P, Maruschak P, Lapusta Y. (2006). Experimental study of crack growth in a bimetal under fatigue and fatigue-creep conditions, International Journal of Fracture 139: 545–552.

[5] Xu S S, Liu S M, Guo S J, Zhang A M. (2009). Use of a submodel method to check the bolt strength of marine equipment, Journal of Marine Science and Application Appl 8: 316-322.

[6] Mian A, Law J. (2010). Geometric optimization of van der Pauw structure based MEMS pressure sensor, Microsystem Technologies 16: 1921–1929.

[7] Liu Y, Shi Y, Zhou Q, Xiu R Q. (2015). A sequential sampling strategy to improve the global fidelity of metamodels in multi-level system design, Structural and Multidisciplinary Optimization.

[8] Nicholas T, Zuiker J R. (1996). On the use of the Goodman diagram for high cycle fatigue design, International Journal of Fracture, 80: 219-235.

[9] Moon S I, Cho I J, Yoon D. (2011). Fatigue life evaluation of mechanical components using vibration fatigue analysis technique, Journal of Mechanical Science and Technology, 25: 631-637.

[10] Xin Z F, Zuo Z X, Feng H H, Neild S A. (2012). Higher order accuracy analysis of the second-order normal form method, Nonlinear Dynamics, 70: 2175–2185.