Finite Element Method Optimization of a Robot Structure

Wang Shijun, Zhao Jinjuan
Department of Mechanical Engineering, Xi’an University of Technology
Xi’an 710048
Shaanxi Province, People’s Republic of China

Abstract
In optimal design for robot structures, design models need to be modified and computed repeatedly. Because modifying usually cannot automatically be run, it consumes a lot of time. This paper gives a method that uses APDL language of ANSYS 5.5 software to generate an optimal control program, which make optimal procedure run automatically and optimal efficiency be improved.

Introduction
Industrial robot is a kind of machine, which is controlled by computers. Because efficiency and maneuverability are higher than traditional machines, industrial robot is used extensively in industry. For the sake of efficiency and maneuverability, reducing mass and increasing stiffness is more important than traditional machines in structure design of industrial robot.

A lot of methods are used in optimization design of structure. Finite element method is a much effective method. In general, modeling and modifying are manual, which is feasible when model is simple. When model is complicated, optimization time is longer. In the longer optimization time, calculation time is usually very little, a majority of time is used for modeling and modifying. It is key of improving efficiency of structure optimization how to reduce modeling and modifying time.

APDL language is an interactive development tool, which is based on ANSYS and is offered to program users. APDL language has typical function of some large computer languages. For example, parameter definition similar to constant and variable definition, branch and loop control, and macro call similar to function and subroutine call, etc. Besides these, it possesses powerful capability of mathematical calculation. The capability of mathematical calculation includes arithmetic calculation, comparison, rounding, and trigonometric function, exponential function and hyperbola function of standard FORTRAN language, etc. By means of APDL language, the data can be read and then calculated, which is in database of ANSYS program, and running process of ANSYS program can be controlled.

Figure 1 shows the main framework of a parallel robot with three bars. When the length of three bars are changed, conjunct end of three bars can follow a given track, where robot hand is installed. Core of top beam is triangle, owing to three bars used in the design, which is showed in Figure 2. Use of three bars makes top beam nonsymmetrical
along the plane that is defined by two columns. According to a qualitative analysis from Figure 1, Stiffness values along z-axis are different at three joint locations on the top beam and stiffness at the location between bar 1 and top beam is lowest, which is confirmed by computing results of finite element, too. According to design goal, stiffness difference at three joint locations must be within a given tolerance. Inconsistency of stiffness will have influence on the motion accuracy of the manipulator under high load, so it is necessary to find the accurate location of top beam along x-axis.

![Figure 1](image)

To the questions presented above, the general solution is to change the location of the top beam many times, compare the results and eventually find a proper position. The model will be modified according to the last calculating result each time. It is difficult to avoid mistakes if the iterative process is controlled manually and the iterative time is too long. The outer wall and inner rib shapes of the top beam will be changed after the model is modified. To find the appropriate location of top beam, the model needs to be modified repetitiously.

This paper gives an optimization solution to the position optimization question of the top beam by APDL language of ANSYS program. After the analysis model first founded, the optimization control program can be formed by means of modeling instruction in the log file. The later iterative optimization process can be finished by the optimization control program and do not need manual control. The time spent in modifying the model can be decreased to the ignorable extent. The efficiency of the optimization process is greatly improved.

**Construction Of Model For Analysis**

The structure shown in Figure 1 consists of three parts: two columns, one beam and three driving bars. The columns and beam are joined by the bolts on the first horizontal rib located on top of the columns as shown in Figure 1. Because the driving bars are substituted by equivalent forces on the joint positions, their structure is ignored in the model.
The core of the top beam is three joints and a hole with special purpose, which cannot be changed. The other parts of the beam may be changed if needed. For the convenience of modeling, the core of the beam is formed into one component. In the process of optimization, only the core position of beam along x-axis is changed, that is to say, shape of beam core is not changed. It should be noticed that, in the rest of beam, only shape is changed but the topology is not changed and which can automatically be performed by the control program.

In Figure 1, six bolts join the beam and two columns. The joint surface cannot bear the pull stress in the non-bolt joint positions, in which it is better to set contact elements. When the model includes contact elements, nonlinear iterative calculation will be needed in the process of solution and the computing time will quickly increase. The trial computing result not including contact element shows that the outside of beam bears pulling stress and the inner of beam bears the press stress. Considering the primary analysis object is the joint position stiffness between the top beam and the three driving bars, contact elements may not used, but constructs the geometry model of joint surface as Figure 2 showing. The upper surface and the undersurface share one keypoint in bolt-joint positions and the upper surface and the under surface separately possess own keypoints in no bolt positions. When meshed, one node will be created at shared keypoint, where columns and beam are joined, and two nodes will be created at non-shared keypoint, where column and beam are separated. On right surface of left column and left surface of right column, according to trial computing result, the structure bears press stress. Therefore, the columns and beam will share all keypoints, not but at bolts. This can not only omit contact element but also show the characteristic of bolt joining. The joining between the bottoms of the columns and the base are treated as full constraint. Because the main aim of analysis is the stiffness of the top beam, it can be assumed that the joint positions bear the same as load between beam and the three driving bars.

![Figure 2](image-url)
The structure is the thin wall cast and simulated by shell element shell63. The thickness of the outside wall of the structure and the rib are not equal, so two groups of real constant should be set. For the convenience of modeling, the two columns are also set into another component. The components can create an assembly. In this way, the joint positions between the beam core and columns could be easily selected in the modifying the model and modifying process can automatically be performed.

Analysis model is showed Figure 3. Because model and load are symmetric, computing model is only half. So the total of elements is decreased to 8927 and the total of nodes is decreased to 4341. All elements are triangle.

![Figure 3](image)

**Optimization Solution**

The optimization process is essentially a computing and modifying process. The original design is used as initial condition of the iterative process. The ending condition of the process is that stiffness differences of the joint locations between three driving bars and top beam are less than given tolerance or iterative times exceed expected value. Considering the specialty of the question, it is foreseen that the location is existent where stiffness values are equal. If iterative is not convergent, the cause cannot be otherwise than inappropriate displacement increment or deficient iterative times. In order to make the iterative process convergent quickly and efficiently, this paper uses the bisection searching method changing step length to modify the top beam displacement. This method is a little complex but the requirement on the initial condition is relatively mild.

The flow chart of optimization as follows:

1. Read the beam model data in initial position from backup file;
2. Modify the position of beam;
3. Solve;
4. Read the deform of nodes where beam and three bars are joined;
5. Check whether the convergent conditions are satisfied, if not, then continue to modify the beam displacement and return to 3, otherwise, exit the iteration procedure.
6. Save the results and then exit.

The program's primary control codes and their function commentaries are given in it, of which the detailed modeling instructions are omitted. For the convenience of comparing
with the control flow, the necessary notes are added.
/BATCH    the flag of the batch file in ANSYS
RESUME,,robbak,db,,0  read original data from the backup file robbak.db
/PREP7    enter preprocessor
      delete the joint part between beam core and columns
      move the core of the beam by one step length
      apply load and constraint on the geometry model
      meshing the joint position between beam core and columns
FINISH    exit the preprocessor
/SOLU     enter solver
SOLVE     solve
FINISH    exit the solver
/POST1    enter the postprocessor
*GET,front,NODE,2013,U,Z  read the deformation of first joint node on beam into parameter front
*GET,back,NODE,1441,U,Z    read the deformation of second joint node on beam into parameter back
lastdif=1       the absolute of initial difference between front and back last time
flag=-1        the feasibility flag of the optimization results
step=0.05      the initial displacement from initial position to the current position
*DO,I,1,10,1     the iteration procedure begin, the cycle variable is I and its value range is 1~10 and step length is 1
dif=abs(front-back)     the absolute of the difference between front and back in the current result
*IF,dif,LE,1.0E-6,THEN    check whether the absolute difference dif satisfies the request or not
flag=1     yes, set flag equal to 1
*EXIT     exit the iterative calculation
*ELSEIF,dif,GE,lastdif,THEN  check whether the dif value becomes great or not
flag=2          yes, set flag 2
      modify step length by bisection method
      perform the next iterative calculation, use the last position as the current position and modified last step length as the current step length
*ELSE        if the absolute of difference value is not less than expected value and become small gradually, continue to move top beam
      read the initial condition from backup file
      enter the preprocessor
AGEN, ,P51X, , ,step, , ,1 move the core of the beam by one step length
      modify the joint positions between beam core and column
Most of the control program above is copied from log file, which is long. The total of lines is up to about 1000 lines. Many codes such as modeling and post-process codes are used repeatedly. To make the program construct clear, these instructions can be made into macros, which are called by main program. This can efficiently reduce the length of the main program. In addition, modeling instructions from log file includes lots of special instructions that are only used under graphic mode but useless under batch mode. Deleting and modifying these instructions when under batch mode in ANSYS can reduce the length of the file, too.

In the program above, the deformation at given position is read from node deformation. In meshing, in order to avoid generating bad elements, triangle mesh is used. In optimization, the shape of joint position between columns and beam continually is changed. This makes total of elements different after meshing each time and then element numbering different, too. Data read from database according to node numbering might not be data to want. Therefore, beam core first needs to be meshed, then saved. When read next time, it’s numbering is the same as last time.

Evaluating whether the final result is a feasible result or not needs to check the flag value. If only the flag value is 1, the result is feasible, otherwise the most proper position is not found. The total displacement of top beam is saved in parameter step. If the result is feasible, the step value is the distance from initial position to the most proper position. The sum of iterative is saved in parameter I. According to the final value of I, feasibility of analysis result and correctness of initial condition can be evaluated.

**Optimization Results**

The sum of iterative in optimization is seven, and it takes about 2 hour and 37 minutes to find optimal position. Figure 4 shows the deformation contour of the half-construct. In Figure, the deformations in three joints between beam and the three driving bars is the same as level, and the corresponding deformation range is between -0.133E-04 and -0.115E-04m, the requirement of the same stiffness is reached. At this time, the position of
beam core along x-axis as shown in Figure 1 has moved -0.71E-01m compared with the original designed position.

![Figure 4](image)

Because the speed of computer reading instruction is much faster than modifying model manually, the time modifying model can be ignored. The time necessary for optimization mostly depends on the time of solution. Compared with the optimization procedure manually modifying model, the efficiency is improved and mistake operating in modeling is avoided.

**Conclusion**

The analyzing result reveals that the optimization method given in this paper is effective and reaches the expected goal. The first advantage of this method is that manual mistakes do not easily occur in optimization procedure. Secondly, it is pretty universal and the control codes given in this paper may be transplanted to use in similar structure optimization design without large modification. The disadvantage is that the topology structure of the optimization object cannot be changed. The more the workload of modifying the model, the more the advantages of this method are shown. In addition, the topology optimization function provided in ANSYS is used to solve the optimization problem that needs to change the topology structure. The better optimization results can be achieved if the method in this paper combined with it.
References