Using ADAMS–simulation for predicting dynamic force histories for the fatigue analysis of a log crane

Aki Mikkola, Heikki Handroos and Erkki Niemi
Lappeenranta University of Technology
Department of Mechanical Engineering
Box 20
FIN–53851 Lappeenranta
Finland
Phone +358–53–621 2439
Fax +358–53–621 2499
E–mail Aki.Mikkola@lut.fi

ABSTRACT

The present paper highlights the fatigue life prediction of a welded log crane structure. The fatigue damage of the log crane is investigated using ADAMS in conjunction with ANSYS finite element software. The coupled dynamics of a hydraulic driven log crane is investigated using ADAMS. The complete simulation model is created by combining the flexible mechanism model with equations describing the hydraulic system. The structure under investigation consists of three hydraulic cylinders and four flexible members of mechanism.

The results of the dynamic analysis, which include individual member velocities, accelerations, joint and applied forces, are printed out at each solution step. The output information of ADAMS is used in the ANSYS program to determine a multiple load–case. Load history is calculated at each time step as separate static analysis. During the dynamic analysis the stress range occurrences are calculated using the Rainflow–algorithm. The fatigue life at a certain location of structure is finally calculated by means of the fracture mechanics approach. In order to investigate the accuracy of the stress results, a comparison between the model and the real structure is carried out.

1 INTRODUCTION

Hydraulic driven cranes are made by welding different components together. Welded joints contain initial cracks and therefore welded structures are disposed to fatigue damage. Fatigue damage occurs due to crack propagation. Crack propagation decreases the ability of the structure to withstand future load and it may lead to final fracture. Crack propagation is caused by the cumulative effect of all stress ranges occurring during the design life.

Investigation of fatigue loads requires that all the subsystems of the crane, such as the electronic control system, the hydraulic system and mechanism, must be studied simultaneously. This can be done by measuring prototypes or using computer simulation. In the present paper investigation of coupled dynamics is carried out by employing ADAMS dynamic system simulation software in conjunction with ANSYS finite element software. The structure under investigation is a three–degrees of freedom commercial log crane.

2 LOG CRANE UNDER INVESTIGATION

The structure under investigation is PATU 655 log crane manufactured by KESLA company in Finland. The crane is suitable for mid–weight tractors and it is actuated by a lift–, swing–, telescope– and turn cylinder. The cylinders are controlled by an electrically actuated 3/4–directional valve, which is connected by hoses to the cylinders. Counter balance valves are included to the hydraulic circuit to prevent the load from running rabidly.

The real crane structure was measured in LUT (Lappeenranta University of Technology) steel structure laboratory using several different sensors. The physical properties obtained by the measurements were the strain of boom, stroke of cylinders and hydraulic pressure in several points of hydraulic circuit. The strain was measured by a single gauge, which was glued in position A in figure 1. The crane with hydraulic circuit is presented in figure 1.

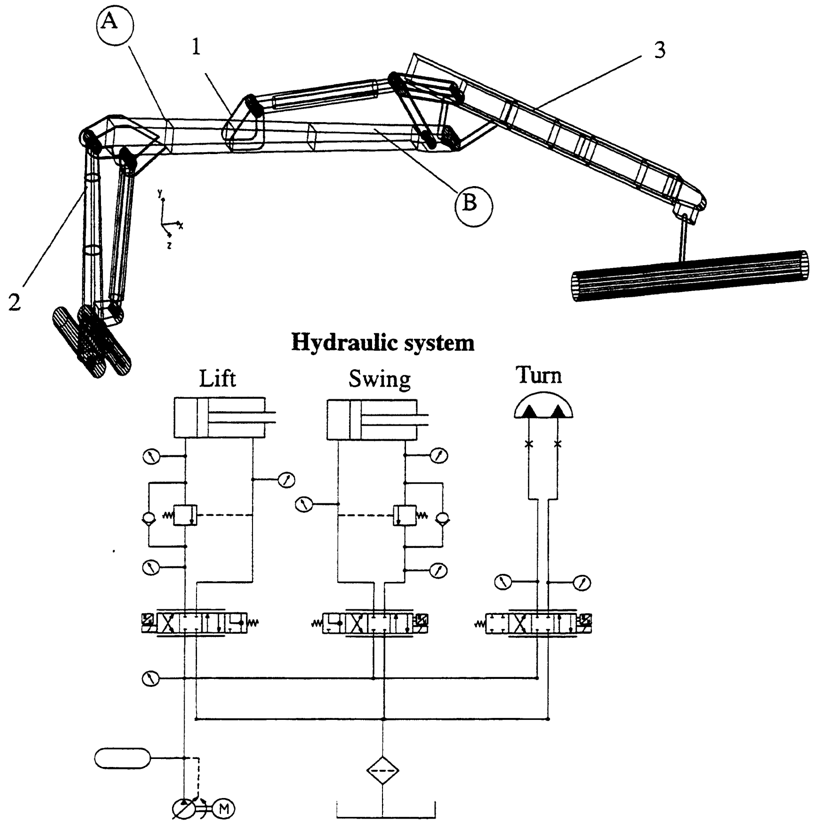

Figure 1 The log crane under investigation

2.1 SIMULATION MODEL OF THE CRANE UNDER INVESTIGATION

The crane is modelled as a flexible mechanism where lift arm, swing arm and pillar are elastic. Moreover the crane connection to the ground is modelled with torsional spring. The spring constant of the ground is obtained measuring the real structure. The boom is modelled flexible by transfer of elastic characteristics from the ANSYS finite element program. Due to the simplicity of the boom's structure, it is modelled using linear 6–DOF beam elements.

The complete simulation model is constructed by combining the flexible mechanism model and equations describing the hydraulics. The hydraulic system is described using fundamental continuity (1) and steady–state flow (2) equations for hydraulic components [3]

$$\dot{p} = \frac{B_e}{V_t} \cdot \left(\sum_{i=1}^{n} Q_{in,i} - \sum_{j=1}^{m} Q_{out,j} \right) \tag{1}$$

$$Q = C_v \sqrt{\Delta p} \tag{2}$$

In equation 2 ΣQ_{in} and ΣQ_{out} describe the sums of flows from and in to the volume. The flows of components can be calculated using steady–state flow equation (2). C_v in Eq. (3) is either a parameter or variable, whose value depends on the system variables or external input signals. The value C_v includes the cross section area of orifice. In the present paper C_v is obtained using semi–empirical approach. Amore detailed description of valve models and the semi–empirical approach can be found in [3], [4], [5] and [6].

To ensure the validity of the simulation model, the calculated results are compared with those obtained by measuring the real structure. This is carried out by moving the lift–, swing– and turn cylinders simultaneously using a pulse input signal for directional valves. Figure 2 represents pressure rates in both chambers of the lift– (A), swing– (B) and turn cylinders (C). Furhermore, figure 2 D represents cylinder stroke during the motion. It can be concluded from figure 2 that relatively good agreement between simulated and measured results is achieved.

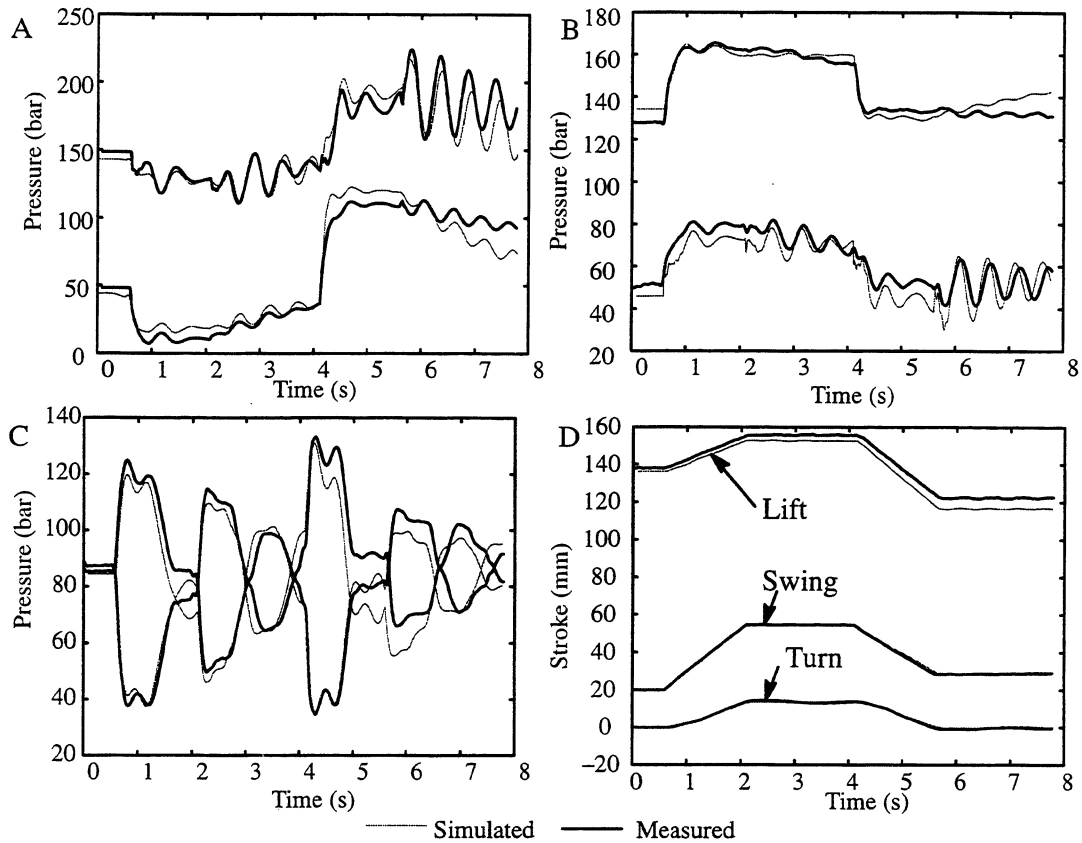

Figure 2 Comparison between simulated and measured results

3 DYNAMIC FORCE HISTORY OBTAINMENT BY UTILIZING SIMULATED RESULTS

Stresses applied to the structure during the simulation are easiest to obtain using stress analysis software, such as the ANSYS finite element program. The results of dynamic analysis, which include individual member velocities, accelerations, joint and applied forces, can be utilized to create loads for a FEM–model. For a rigid member of the mechanism the loads can be created by using ADAMS/FEA translation toolkit. Unfortunately ADAMS/FEA translation toolkit can not be used for a flexible member of the mechanism. For that purpose a program has been developed , which reads velocities and accelerations of each mass point of the flexible member of the mechanism from the results file and converts them to nodal forces. The program also reads external forces applied to the structure and combines them together with inertial forces. Stresses applied to the structure are reasonable to compute at each simulated time step. This is because there are a number of local minimum and maximum stresses and it is time consuming to find them out. These stresses are caused by combinations of different force components.

Stress history can be calculated at each time step as a separate static analysis by joining multiple load cases together in a finite element model. The multiple stress analysis is not time consuming since the stiffness matrix is created only once. Stress history of several critical points can be printed out as a result of the stress analysis. Figure 3 presents calculated and measured stress history in point A (fig. 1) during the verification simulation.

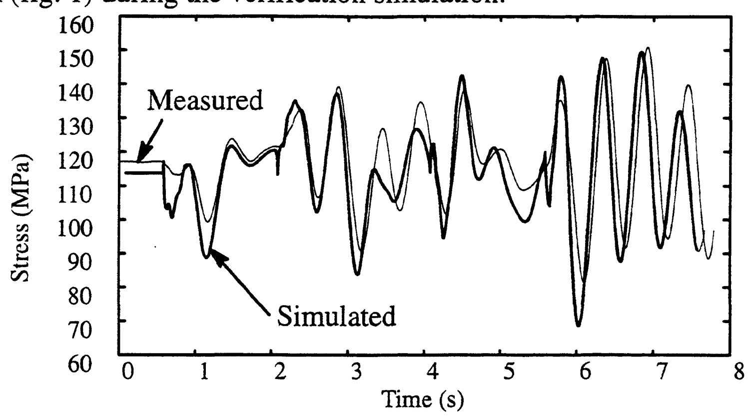

Figure 3 Calculated and measured stress history in the point under investigation

4 EXAMPLE OF FATIGUE LIFE PREDICTION OF LOG CRANE

Stress ranges applied during the simulation are required for fatigue analysis. Stress range occurrences during the dynamic analysis are preferably calculated using the Rainflow procedure, which is based on the cycle–counting method. The rainflow procedure stores the stress history of measurement or simulation in the form of ranges [4]. Experimental fatigue test are mainly made at constant amplitude stress. These results can be utilized by calculating equivalent constant amplitude stress from variable amplitude stress. Equivalent stress replaces several different sizes of stress ranges by one equivalent stress range. The fatigue life can be calculated by employing the nominal stress approach, structural hot spot stress or strain approach, local notch stress approach, and fracture mechanics approach. In the present example the fatigue life is calculated using the fracture mechanics approach.

To highlight fatigue life the prediction behavior of the log crane is simulated during a typical work cycle. At the beginning of a work cycle the crane is moved near to a log and the log is connected to the end of crane. The loaded crane is lifted to a certain position by using the lift–, swing– and turn cylinder. Finally the log is moved downwards using the lift– and swing cylinder and at the end of the work cycle the log is released. Figure 4 A presents stress history at point A (fig 1) during the work cycle. Near position A there is a welded round nut used for fastening hydraulic pipes. The round nut is investigated in [2] and it has been discovered to be the most crucial point in terms of fatigue. Figure 4 shows that heavy vibration occurs during the deceleration and acceleration of loaded crane. Furthermore components of the three biggest ranges, obtained by the Rainflow procedure, are marked in figure 4. Figure 4 b represent the effect of different size of stress ranges to the total fatigue failure. Figure 4 b is obtained using fracture mechanism approach and it shows that the biggest stress range causes over 50 % of the overall fatigue failure. According to the fracture mechanism the fatal fatigue failure will occur after 54 600 work cycles.

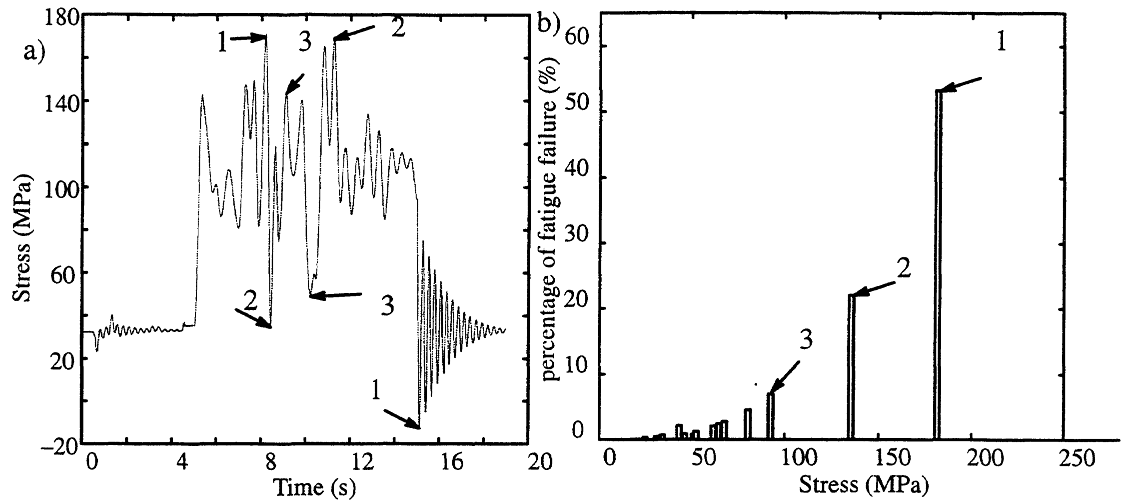

Figure 4 a) Stress history during the simulation. b) Component of fatigue failure

5 CONCLUSION

The present paper shows how to obtain the stress history and the fatigue life of a hydraulic actuated log crane. The coupled dynamics of the hydraulic driven log crane is simulated using ADAMS. The complete simulation model is constructed by combining the flexible mechanism model with equations describing the hydraulic system. After analyzing the dynamics of the system the results obtained are utilized in the ANSYS finite element program to determine the stress history of the boom. In order to investigate the accuracy of the simulation model a comparison between the model and the real structure is carried out.

ADAMS program in conjunction with ANSYS provides an effective way to determine stress history applied to the structure, and fatigue damage at critical locations of the structure. Integrated use of these programs makes it possible to investigate the effect of different design parameters to the fatigue of structure. However the use of these programs is restricted for quite short period work cycles due to the limit of computer capacity. Also different kinds of statistical phenomena, such as different drivers and different environment conditions can not be taken into account.

REFERENCES

1. Niemi. E., Stress determination for fatigue analysis of welded components, IIS/IIW–1221–93, Abington Publishing, 1995.

2. Partanen T., Tarjavuori P., Niemi E., Definition of hot spot stress gradient in stiffened plate structures based on the beam on elastic foundation theory. IIW Doc. XIII –1478–92

3. Handroos, H.M., Keskinen E.K., Vilenius M.J., Effect of design parameters on vibrational behavior of a counter balance valve equipped hydraulic cranes. 2nd JHPS International Conference on Fluid Power, Tokyo, September, 1993

4. Watton, J., Fluid power systems: modeling, simulation, analog and microcomputer control, 1989 Prentice Hall International (UK) Ltd

5.Handroos, H.M, A method for postulating flexible models for individual components for the fluid power circuit simulation, ASME, FLUCOM'91, San Francisco, August 29–31

6. Rouvinen A., Mikkola A. Handroos H., Coupled simulation of a hydromechanical system. 10th European ADAMS Users' Conference. Frankfurt/Niederrad Germany, November 14th–15th, 1995