

2014

BioTechnology

*An Indian Journal***FULL PAPER**

BTAIJ, 10(11), 2014 [5584-5589]

Finite element analysis of medium-wheel Bulldozer earth-moving shovel arm based on ANSYS

Qian YinCollege of Engineering, Xi'an International University, Xi'an Shaanxi 710077,
(CHINA)

ABSTRACT

The working system acts as an important component, its performance and reliability are the main criteria to measure the comprehensive function of a Bulldozer. Since Bulldozers are used in difficult conditions, it is required that the earth-moving shovel arms should be strong and hard enough. If calculated by the general mechanics of material, the reliability of its stress can not be accurate. With the development of engineering and computer technology, the complicated approach of finite element has come to be widely employed. It has laid foundation for the further design of the earth-moving shovel, setting up a module by employing. Three-dimensional software Pro/E first, then analysing and calculating its stress by ANSYS software to know its stress and deformation state.

KEYWORDS

Finite element; Bulldozers; Earth-moving shovel arm.



INTRODUCTION

With the development of science and technology, the finite element approach has been widely employed. By employing the finite element approach, the stress and strain analysis of the various structures with geometrical shapes can be possible. ANSYS is a commonly-used analysis software of finite element, a blend of structure mechanics analysis, non-linear analysis, thermal analysis, Electromagnetic analysis and hydromechanics analysis. Besides, it has five more advanced functions, namely, coupling effect of multi-physics field, optimization design, topology optimization, element birth and death; Unified Power Format (UPF). Besides, ANSYS provides its users with three methods to generate geometric and finite-element models, one is generating finite element directly; the second is creating geometric models, dividing network in ANSYS and then generating finite element; the third is importing entity or finite-element models made in other CAD systems. If an entity model is to be imported, it has to be processed before dividing network and generating finite-element model^[1]. The paper, employing the third method, sets up an Earth-moving shovel arm model of a medium wheeled-Bulldozer and, taking its working condition into full account, analyzing its stress and strain with the help of ANSYS software in order to detect any weakness.

ANALYZING THE 3-D MODEL OF EARTH-MOVING SHOVEL ARM AND FINITE ELEMENT

Building and analyzing of 3-D models of earth-moving shovel arm

Bulldozers are widely used in many industries such as the construction of mines, municipal projects, road works. Earth-moving shovel arm serves as the working part of a Bulldozer and works in difficult conditions, technical requirements of part structure, material, and manufacturing technology are quite high. In most cases, it will be rather complex to build geometric models for irregular line, surface and bodies in ANSYS. Therefore, build a geometric model in Pro/E software first, then import it into ANSYS by the interface Connection for Pro/Engineer, which the ANSYS has for Pro/E. Connection for Pro/Engineer can directly convert Pro/E model data to ANSYS, meanwhile it has the function of designing parameter optimization on the basis of executive components^[2]. Get the feasible model after processing and geometrically simplify the details of the entity model and then accurately divide networks and compute.

Set design requirements:

Rated power of Bulldozers:120KW (medium);

Installation: rotary;

Operation: hydraulic pressure;

Depth of cutting: 400mm.

The main parameters obtained by applying analogy and structural parameters of wheeled-bulldozers DL210G are as follows (Figure 1): $l_g=2020\text{mm}$, $l_1=1125\text{mm}$, $l_2=493\text{mm}$, $t=20\text{mm}$, $k=100\text{mm}$, $h=180\text{mm}$.

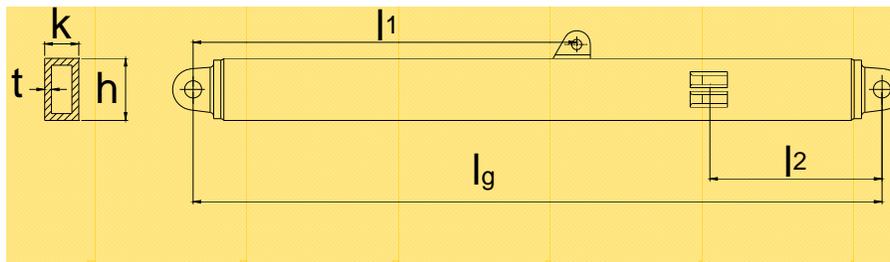


Figure 1 : Structure chart of earth-moving shovel arm

The 3-D model of earth-moving shovel arm assembled by employing Pro/E software (Figure 2), has left and right arms like boxes, which are light, strong and rigid enough to fully meet the working requirements. With ANSYS software applicable to many engineering models, link Pro/E to ANSYS, process and analyze the three-dimensional model of earth-moving shovel arm (Figure 3)^[3].

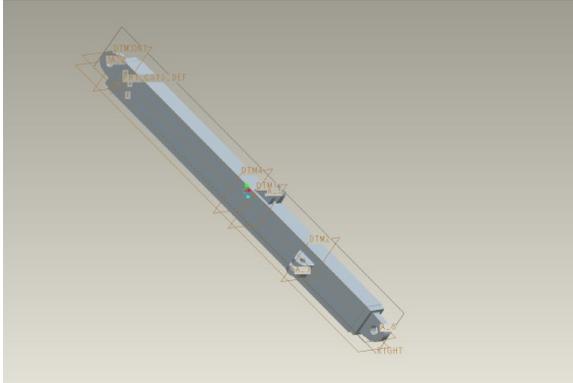


Figure 2 : Model of earth-moving shovel arm

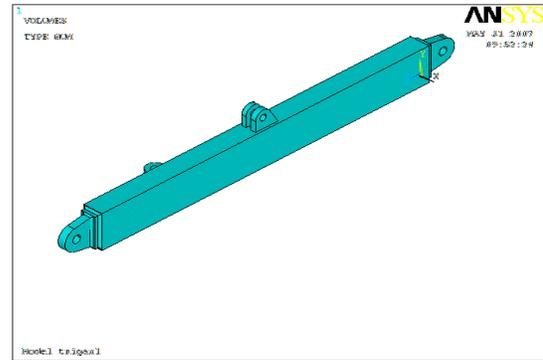


Figure 3 : The importing icon

Definition of finite element of earth-moving shovel arm

Analyze the finite elements of the same structure, with different unit types and unit attribute for different analytical types. Different unit types are processed by different analytical types and different material. Besides, several attributes are available for many units to accomplish different function. Once the unit is selected, the physical environment in which to address the issue can be certain.

Choose Solid45 unit in the model, a square normally used for dividing 3-D entity unit. Use Solid45 unit to build 3-D entity structure of a earth-moving shovel arm, 8 nodes will be used to define it with 3 degrees of freedom of translation along xyz directions. The unit has such properties as plasticity, creeping, swelling, stress stiffening, large deflection and strain. Earth-moving shovel arms, normally made of carbon structural steel, here in the study choose to use Q235^[4], the elastic Modulus of EX input material in the ANSYS is 2.06E11, the Poisson's ratio of PRXY input material is 0.3.

Grid division of earth-moving shovel arm

There exist reliability and validity of computing with finite element approach, the same as approximate-number analytical methods. The result errors of finite element analysis can come from any step of the analyzing process, discretization is one of the main reasons. How well the grid of finite element is divided has an important influence on the precision of the result of analysis.

Grid division between earth-moving shovel and the arm is Main Menu> Preprocessor> Meshing> Size Cntrl> Lines> All Lines^[5]. The computing of simplified frame does not need too many grids (The number of grids has little influence on the computing results), so four is enough for each beam.

No. of element division enter 4.

Main Menu> Preprocessor> Meshing> mesh> Lines> pick All

Generates simplified model node and unit map, as shown in Figure 4. In this study breaks take place on the section or border with the help of the optimizing function of the software, hence here grid division is fairly delicate while rather rough at the places with few and gentle sections. The division result is as shown in Figure 5.

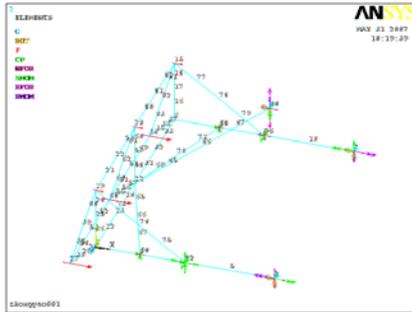


Figure 4 : Node, unit of simplified model and Load on rendering

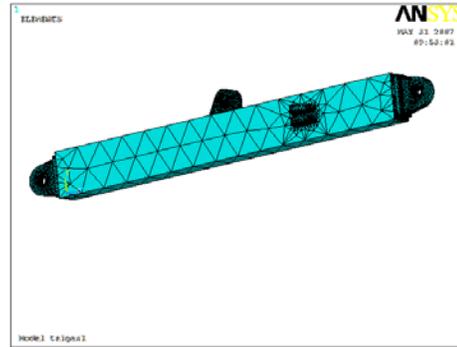


Figure 5 : Grid division

Exerting constraint and load on the model of earth-moving shovel arm

Normally the load in finite element analysis includes boundary condition and the effect the interior and exterior environments have on the object, including degrees of freedom, concentrated load, side load, load and inertia load. Load can be applied on entity or finite element models. If applied directly on entity model, because it is independent from the division of finite element grid, load applying will not be influenced by any new division or partial change of grid. Moreover, it is easy to operate and can be collected in the model. But for either of the two ways, the analyzing procedure of finite element will be transformed to the model of finite element before the final computation. Therefore, the load applied on entities will automatically change to the node units where it belongs to.

Treating the earth-moving shovel arm as a beam and analyzing it with the two points fixed, choose hole surface of connection pin and then choose All DOF to exert constraint.

Choose the middle hole surface, when entering Value for the first time, in the analytical results of the simplified model, at node 3 and node 4 (Figure 4) load is exerted in three directions (The load is obtained from the assembled design of earth-moving shovel and the place with heavier load is chosen).

Analysis of computing results of earth-moving shovels

After exerting load, deformation happens in earth-moving shovels, as shown in Figure 6. It can be seen from the figure that the most serious deformation takes place at the center of the arm but slightly upper, namely, the hinge joint.

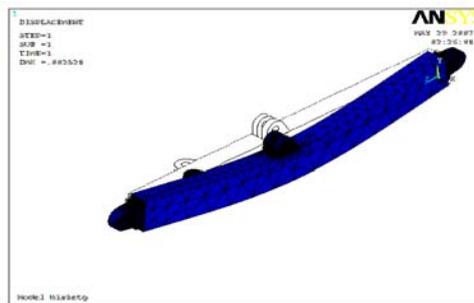


Figure 6 : Deformation of earth-moving shovel after applying load

Checking equivalent stress nephogram (Figure 7, Figure 8): in the equivalent stress nephogram, as shown by color column scale, blue indicates the least stress and red the most It can be seen from the nephogram that there is more stress near the four boring. And the fourboring are easily broken in actual work.

$SMX = 97.005MPa$.

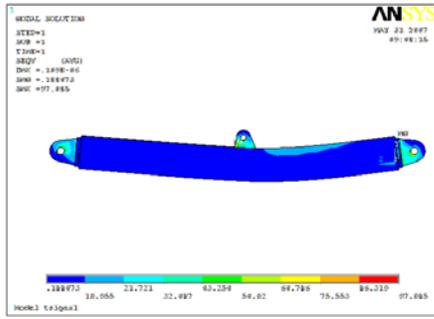


Figure 7 : The stress nephogram

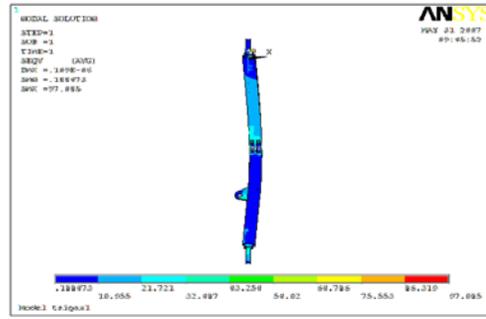


Figure 8 : The stress nephogram

Checking the node displacement variation (Figure 9, Figure 10): As shown in the figure, blue means the least displacement of the node and red the most. The largest deformation of the arm:

$DMX = 0.389 \times 10^{-6} m$.

Since the deformation of arm displacement is not obvious, it can be known according to the experience that the displacement will not lead to instability of earth-moving shovel arm.

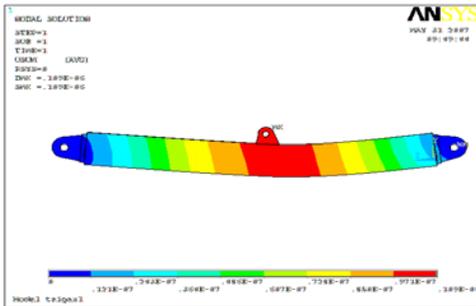


Figure 9 : Displacement of node

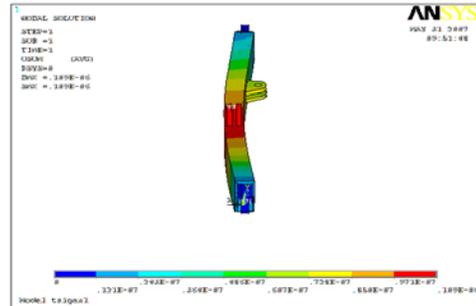


Figure 10 : Displacement of node

With the analysis and computation in ANSYS, the stress of all the nodes are as follows:
General Postprocessing>List Results>Nodal Solution> Stress>von Mises stress

PRINT S NODAL SOLUTION PER NODE					
***** POST1 NODAL STRESS LISTING ***** (unit: MPa)					
9665	8.2328	-8.2812	-12.179	20.412	18.769
9666	13.283	-6.8429	-11.490	24.773	22.807
9667	14.641	-4.1703	-9.8728	24.514	22.218
9668	14.373	-4.8043	-7.0332	21.406	20.383
9669	8.4980	-1.2385	-5.4523	13.950	12.393
9670	11.500	-1.3380	-5.4616	16.961	15.322
MINIMUM VALUES (unit: MPa)					
NODE	2961	7895	7999	599	599
VALUE	-16.413	-27.321	-77.515	0.21796	0.19171
MAXIMUM VALUES (unit:MPa)					

NODE	4290	4290	4290	7926	7926
VALUE	91.315	43.745	18.622	96.747	84.227

As can be seen from the results in ANSYS software, the stress is the most at node 7926 and it is 96.747Mpa.

STRENGTH CHECK OF EARTH-MOVING SHOVEL ARM^[6]

Q235 the minimum strength of extension

$$\sigma_s = 225MPa$$

$$\sigma_{\max} = 96.747MPa < \sigma_s$$

$$S = \frac{\sigma_s}{\sigma_{\max}} = 2.3$$

The maximal stress of earth-moving shovel arm is far less than the minimum of Q235 strength of extension, and the safety factor of its structural design has certain safety margin, so it can meet the requirement of life circle for an earth-moving shovel arm to be used in normal conditions.

CONCLUSION

This paper, applying ANSYS software and Pro/E 3-D model, made a finite element analysis of earth-moving shovel arm. It deals with the most fragile part of an earth-moving shovel arm and is consistent with the actual damage. The result can be compared with the material strength since it can be directly known due to the advanced computational function of the software. In this way the usual abundant computation can be saved and thus offers a reference for production designing.

REFERENCES

- [1] M.Ghazani, A.Vajd; Finite Element Analysis of the Groove Pressing of Aluminum Alloy, Modeling and Numerical Simulation of Material Science, **4(1)**, 32-36 (2014).
- [2] P.Lenka, A.Choudhury; Analysis of trans tibial prosthetic socket materials using finite element method, Journal of Biomedical Science and Engineering, **4**, 762-768 (2011).
- [3] C.Zekios, P.Allilomes, G.Kyriacou; Eigenanalysis of Electromagnetic Structures Based on the Finite Element Method, Applied Mathematics, **4(7)**, 1009-1022 (2013).
- [4] R.Yerrawar, V.Tungikar, S.Gawande; Finite Element Analysis of Dynamic Damper for CV Joint, Energy and Power Engineering, **4(4)**, 241-247 (2012).
- [5] S.Alami, A.Elakkad, J.El-Mekaoui, A.Elkhalfi, M.Hammoumi; A Finite Element Approximation of the Stokes Equations, World Journal of Mechanics, **1(6)**, 299-305 (2011).
- [6] P.Rakesh, V.Sharma, I.Singh, D.Kumar; Delamination in Fiber Reinforced Plastics: A Finite Element Approach, Engineering, **3(5)**, 549-554 (2011).