Design Optimization for Suction Boom Arm

[#1]Sanjay Kale, [#2]G.M. Dhanavade, [#3]Prof. S.C. Shilwant

[#1]M.E. Automotive Engineering Sinhagad Academy of Engineering, Kondhwa Pune, India
[#2]SD.G.M. Design Dept. KAM-AVIDA Enviro Engineers Pvt. Ltd, Hinjewadi Pune-411057, India.
[#3]M.E. Automotive Engineering Sinhagad Academy of Engineering, Kondhwa Pune, India

Abstract— The project deals with the design and analysis of a tank accessory named suction boom. This article aids to comprehend the selection procedure and design of various building blocks of the assembly, its applications, analysis reports, etc. Suction boom is a mounting on the tank of vacuum and jetting machines which can be used for the suction of hazardous wastes viz., solid and liquid wastes, dust, chemicals, sewage etc. The building blocks include leak proof swivel joint, turret, arm and hydraulic cylinder assemblies. It consists of an arm rotating in 300° that will position the hose at required place. Suction boom is driven by a hydraulic motor and power is transmitted using a worm worm-wheel drive. A hydraulic cylinder is so selected to lift the arm in 300mm. Once the hose is positioned to the destination, suction pressure is created and the sewage lines/ piping are then cleaned. Design iterations were carried on and the most preferable de-sign was selected considering parameters like strength, machinability, cost, availability of materials, etc. Once the concept was finalized, the entire assembly was divided into number of sub-assemblies for simplification and then each assembly was worked out for its strength and design/ selection of its individual components (For example: worm and worm wheel, hydraulic cylinder, knife gate valve, bearings, etc.). The turret and arm assemblies were analyzed using CAE tool.

Keywords: Topology, Topography, Natural Frequency, Stress Plots, Displacement Plots, OptiStruct, Hypermesh

I. INTRODUCTION

Liquid waste comprises of chemical waste from industries, effluents, sewage, etc. Let us consider an example of sewage waste handling, its handling include transportation of sewage from its source to the sewage treatment plant. Sewage is water that often contains human waste (feces and urine). A variety of organisms live in the gastrointestinal tract of human beings. Many of these including bacteria, viruses and parasites end up in human waste and can be transmitted to other human being. It also contains certain toxic chemicals which can further form some deadly toxics by combining with other compounds and threaten human health.

Following are the routes that can cause human exposure to the toxic contains of sewage water:

a. Oral: Drinking contaminated water or eating contaminated seafood.
b. Dermal: Getting contaminated water on your skin and open cuts and rashes.
c. Aerosol: Inhaling water droplets such as those from breaking waves.

The most commonly reported illnesses associated with oral and aerosol exposure are gastro-enteritis (Infection of gastrointestinal tract) and hepatitis (Infection of liver). The dermal exposure to sewage may lead to infection of open cuts or rashes. These are just few risks associated to liquid waste handling, it includes for risks which are caused due to industrial effluents, wastes in petrochemical industries, handling chemicals in industrial environments, etc.

The people involved in these activities (for example- waste handling, sewage line cleaning, transportation of waste, etc.) are more prone to the health hazards caused by these toxic wastes.

Suction boom plays a vital role in reducing human exposure to the toxic liquid waste that needs to be handled daily. The human involvement is reduced as the process of handling the waste as well as cleaning the sewage lines becomes mechanized.

II. LITERATURE REVIEW

Suction boom is a mounting on the tank of vacuum and jetting machines which can be used for the suction of hazardous wastes viz., solid and liquid wastes, dust, chemicals, sewage etc.

Due to the traditional approach of cleaning in India, mechanized cleaning is kept at the bay by the government as well as by the private bodies. Suction boom is used for many purposes in various developed countries due complete cleaning mechanization. Suction boom and even other cleaning machines are rarely seen in our country. The suction boom we designed seems to be complex but greater advancements are made in this technology by various international companies.

Jurop Spain, Moro Industries USA. Cappellotto Italy, Umole UK are using the advanced motorized suction boom with light weight and optimized structure.

Digging Simulation Model shown in SAE paper 2013-01-2411 seek an optimized digging trajectory.

This helps to calculate digging reaction forces more quickly than 2D-DEM models. In addition, we can calculate generative force and energy consumptions of hydraulic cylinder bysolving inverse dynamics of front linkage of an excavator.

SAE 2012-01-2040 address some aspects regarding the hydraulic circuit modelling for commercial excavator, analysis of feature of the circuit have been also covered. The PDCV's stack and LS signal generation with LSG line for boom. Future steps are aimed at further modifying the hydraulic circuit by replacing the PDCVs stack in order to predict potential energy savings.

SAE 2012-01-0952 showed the virtual process for the Boom Arm Design optimization. Fatigue life evaluation of a hydraulic excavator based on flexible multi-body system dynamics was proposed. This approach makes it possible to analyze life time of them with least assumption of lading distribution. And the automated virtual design systemis developed for optimum design of attachments considering durability and light-weight.
This helps in reducing the time. This is the future scope we identified to have virtual modelavailable.

SAE 2003-01-3427 paper presents a new design methodology, which uses topological and shapeoptimization to arrive at the most efficient use of material. This method can indicate non-intuitive part configurations and can result in lighter weight parts that still able to meet targets for stress and deflection.

III. SCOPE

1. Design of Boom Arm for weight reduction concept
2. Design of Slewing Bearing
3. Weight optimization with FEA Tool
4. Development of prototype
5. Validating the design on Vehicle

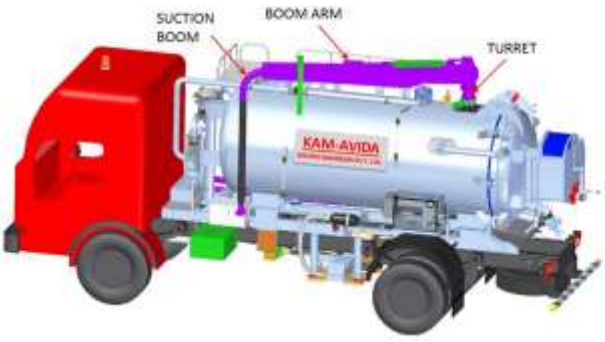

Fig.1 Installation of Boom Arm on Suction Machine

IV. METHODOLOGY

A new design methodology is described to develop three-dimensional CAD structural components using computer aided engineering tools.The proposed method relies on procedures for arriving at the correct load paths in the component and for optimal distribution of material.Finite Element Analysis is done for reducing the interactions

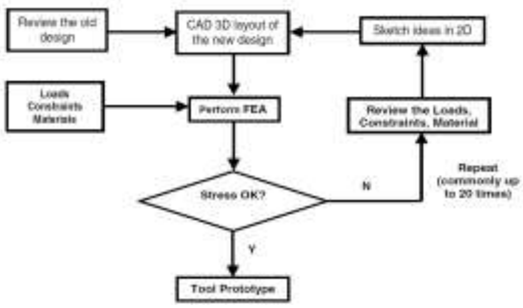

Fig. 2 Old Method to Solve Engineering Problem

CAE (Computer Aided Engg) is a method of rep-resenting and solving any engineering problems by simulating actual working conditions virtually using numerical methods, and computers due higher computational speed. The following table shows various methods of solving any engineering problem:

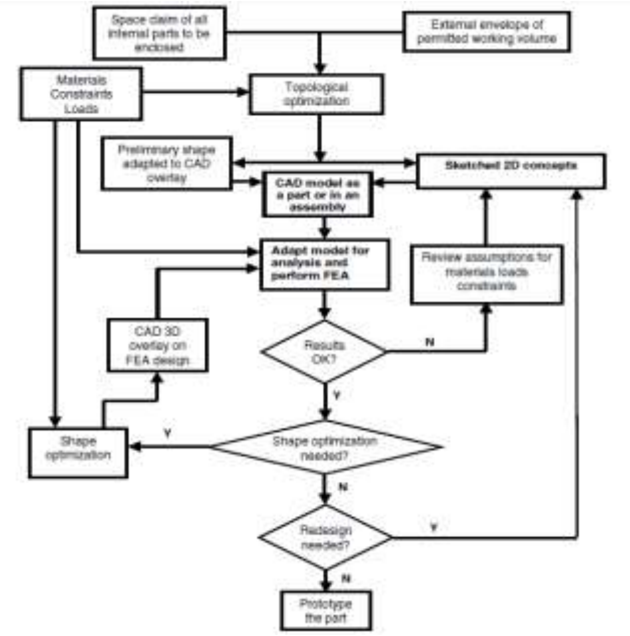

Fig. 3 New Method to Solve Engineering Problem

FEM (FINITE ELEMENT METHOD)

FEM is a numerical method adapted for Mathematical rep-presentation of an actual problem. It is an approximatemethod; the Finite Element Method only makes calculations at a limited (Finite) number of points and then interpolates the results for the entire domain (surface or volume). The Finite Element Analysis (FEA) is a numerical method for solving problems of engineering and mathematical physics. Useful for problems with complicated geometries, loadings, and material properties where analytical solutions cannot be obtained.

ADVANTAGES OF FEM

Better Visualization
Minimized design cycle time Reduced No. of prototypes
Virtual Testing and Optimum design+

Fig.4FEM Processes

4.1. Analysis of Suction Boom

As discussed earlier suction boom is an accessory of a jet-ting machine and is subjected to following types of forces while working or at rest: - 1. Vertical load of pipe and slurry in it while suction also jerks while suction. 2. Self-weight of the arm. 3. Cylinder forces while lifting and lowering of arm. 4. Forces while rotation of arm. 5. Dynamic forces while driving the vehicle, and additional external forces to minimize the complexity of analysis the suction boom was divided into various subassemblies. Such divisions allow easy pre-processing but results may deviate from original values, such considerations are made by using high factor of safety.

ANALYSIS TOOLS USED:

Pre-processor- Hypermesh 12.0 Solver- Radioss 12.0

Post-processor- Hyperview 12.0

The stepwise procedure of analysis and results are dis-cussed below:-

STEP-I: GEOMETRY IMPORT AND EDITING.

The required geometry for analysis is imported into Hypermesh in standard IGES format. The imported geometry is edited in Hypermesh to remove surfaces which are not going to take part in analysis, for example, roller plate, rollers, edge fillets, etc. It is done to improve the meshquality.

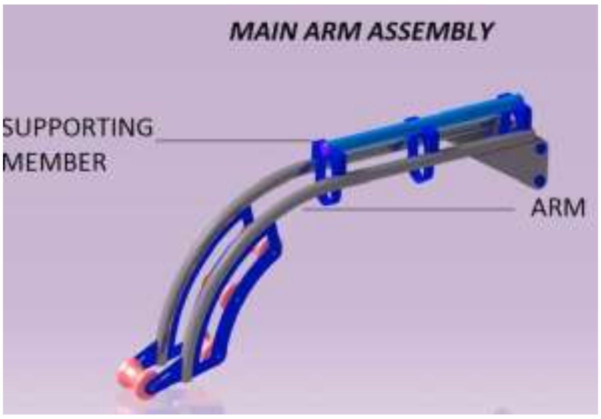

Fig.5 Assembly of Boom Arm

STEP-II: MESHING

Once the geometry is in an appropriate state, a mesh is created to approximate the geometry. A solid 3D mesh is created; with element size of 4 mm is used. Since arm is connected as single enclosed volume welds are also approximated. The specification of mesh is as follows:- element size:- 4mm surface elements:- tries solid elements:- tetras comp.QI:- 0.00

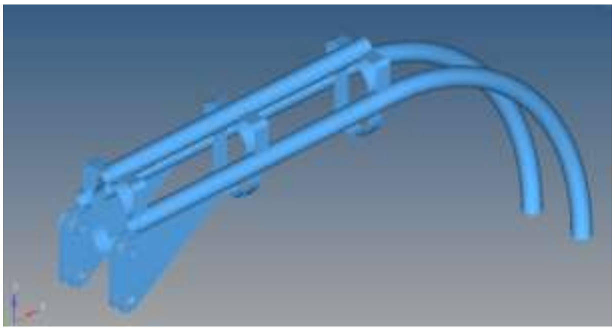

Fig.6 Boom Arm Mesh Model

STEP-III: MATERIAL AND PROPERTY INFORMATION

After meshing is completed, material (e.g. Young’s Modulus) and property information (e.g. thickness values) are assigned to the elements. The following material properties were considered; Material selected:- Steel density of steel:-7850kg/m3 young’s modulus:- 2.1e5 n/mm2 Poisson’s ratio:- 0.3

STEP-IV: LOADS, CONSTRAINTS AND SOLVER IN-FORMATION.

Various loads and constraints are added to the model to represent the loading conditions that the part(s) are subjected to. Different load cases can be defined to rep-resent different loading conditions on the same model. Solver information is also added to tell the solver what kind of analysis is being run, which results to export, etc. The total final load is dividing by no. of selected nodes and is applied on each selected node. The upper arm pin is constrained, whereas lower arm pin is subjected to cylinder actuation force. The loading is done as follows: Weight of arm: - 1500 N Dry weight of hose:-100 N Weight of slurry being carried:- 350 N Unknown downward force:- 50 N Total load:- 2000N, and Factor of safety: - 03 (standard if human involvement). Final load:- 2000 x 3= 6000 N Cylinder force:- 20000 N

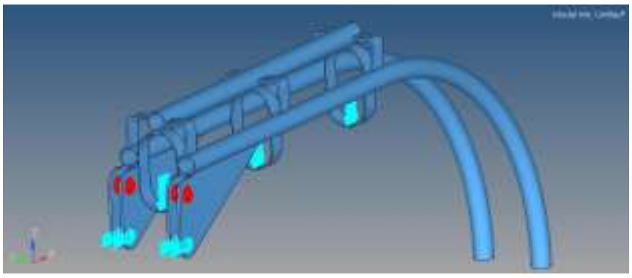

Fig. 7 Boundary Condition and Loading

STEP V: SOLUTION

During the solution phase of a simple linear static analysis or an Eigen frequency study, there is not much to do. The default settings of the Finite Element program do handle these classes of problems pretty well. If the solution process is aborted by an error, it is due to mistakes made during the model building phase. Just to mention a few typical errors: Element quality. Invalid material properties. Material property not assigned to the elements. Insufficiently constrained model (the model shows a rigid body motion due to external loads).

STEP VI: VISUALIZATION / POST-PROCESSING

Once the solution has ended successfully, post-processing (in HyperView for contour plots) of the simulation results is done next. Stresses, strains, and deformations are plotted and examined to see how the part responded to the various loading conditions. Based on the results, modifications may be made to the part and a new analysis may be run to examine how the modifications affected the part. This eventually completes the FEM process. The following are the results of analysis of arm

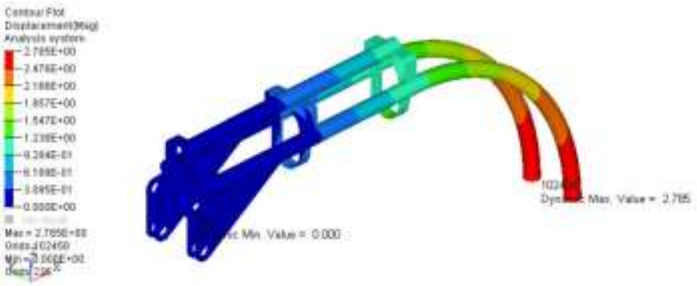

Fig.8 Displacement Plot

Maximum displacement = 2.785 mm

Fig. 9 Von-misses Stress Plot

Maximum stress:- 133.703 N/mm2 ¡ Syt = 370 N/mm2 Hence, Design of Arm is safe.

4.2. Analysis of Turret

Turret connects the arm with the swivel joint, it is mounted on worm wheel for rotation, and it carries the arm in cantilever position and hence is subjected to cylinder force and weight of arm and hose. All the forces on turret are transmitted by arm at two points of connection i.e.

1. Upper arm pin.
2. Cylinder mounting point at pipe mounts plate.

STEP-I: GEOMETRY IMPORT AND EDITING.

The required geometry for analysis is imported into Hypermesh in standard IGES format and the surfaces which are not required are featured with help of various tools available in Hypermesh. Some surfaces are removed to make turret an enclosed volume. For e.g.: edge fillets, surf fillets, small holes, etc.

Fig.10 Mesh Model of Turret

STEP-II: MESHING

After geometry cleanup a solid 3D mesh is created, to discretize the geometry enclosed in single volume. The specification of is mesh achieved using auto mesh panel is as follows: Element size: 4 mm Surface elements: Trias Solid elements: Tetras Comp. QI: 0.005

STEP-III: MATERIAL AND PROPERTY INFORMATION

After meshing is completed, material (e.g. Young's Modulus) and property information (e.g. thickness values) are assigned to the elements. The following material properties were considered; Material selected:- steel. Density of steel:-7850kg/m3 Young's modulus:- 2.1e5 N/mm2 Poisson's ratio:- 0.3

STEP-IV: LOADS, CONSTRAINTS AND SOLVER IN-FORMATION.

The turret is constrained at the bolting point of mounting flange. The forces acting are equally divided and applied on number of nodes which are subjected to load. Since, Turret is heavily loaded to get more accurate analysis, modal analysis is also performed on the turret assembly. The turret was loaded as follows Final load:- 2000 x 3 = 6000 N Cylinder force = 20000 N Eigen frequencies = 200 Hz.

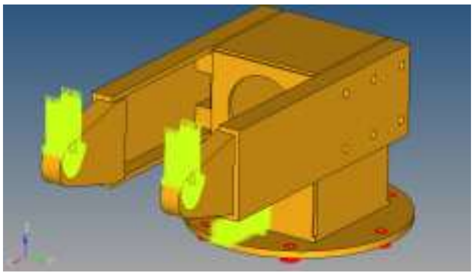

Fig. 11 Boundary Condition and Loading

STEP V: SOLUTION

The generated FEM file is submitted to Radioss for solution of solver deck. Radioss solves the given solver deck and generates a result file which can be read in the Hyperview postprocessor. Radioss can also generate results file for LS Dyna, Ansys, Abaqus, Nastran, etc.

STEP VI: VISUALIZATION / POST-PROCESSING

Once the solution has ended successfully, post-processing (in HyperView for contour plots) of the simulation results is done next. Stresses, strains, and deformations are plotted and examined to see how the part responded to the various loading conditions. The displacement and maximum von-misses stress is plotted to view the results the obtainedvalue is within the yield stress value. The following are the results of analysis of arm.

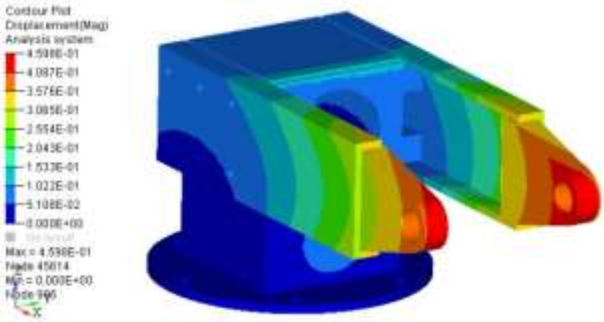

Fig. 12 Displacement Plot

Maximum displacement = 0.46mm

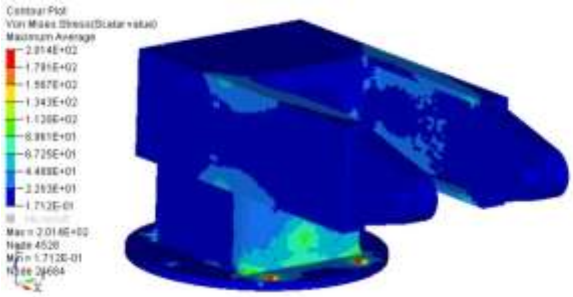

Fig. 13 Stress Plot

Maximum stress:- 201.4 N/mm2 ¡ Syt = 370 N/mm2 Hence, Design of Turret is safe.

V. PROTOTYPE

Prototype Fabricated Boom Arm is made for the reduced weight design as suggested after FEA study.

Fig.14 Prototype of Boom Arm Assembly

VI. RESULTS AND FUTURE WORK

1. Total weight reduction in suction boom = 92 kgs (Old weight of suction boom = 257 kgs. and New weight of suction boom = 165 kgs).
2. Optimization technique resulted in more efficient design path to final CAD model. A non-intuitive shape was devised from the initial topological optimization, which allowed the structure to be more efficient for stiffness versus weight.
3. Bearing Life is increased with new bearing.
4. Motorized Suction Boom is reducing effort for locating suction boom to the target.

Future work identified in Automated Virtual Design System is developed for optimum design of attachments considering durability and light-weight. This helps in reducing the time.

VII. CONCLUSIONS

This paper presents a new design methodology, which uses topological and shape optimization to arrive at the most efficient use of material.

This method can be used to design new structural components where the load system and part geometry are complex and load paths are not obvious. This method can result in lighter weight parts that are still able to meet the targets for stress and deflection. The design technique used was a topological optimization on a free space 3D model as a first step in optimization of the FEA approved design. Results were favorable for overall design time and the final weight and stiffness of the component.

REFERENCES

1. Seong-Yong Kim.,(2012), EA Development of the Virtual Design System for the Attachments of a Hydraulic Excavator, *SAE International*,2012-01-0952.
2. Tkayuki Koizumi, (2013)Digging Trajectory Optimization by Soil Models and Dynamics Models of Excavator, *SAE International*,2013-01-2411.
3. Gabriele Altare,(2012) Analysis, Modelling and Simulation of the Hydraulic Circuit,*SAE International*, 2012-01-2040.
4. Ramesh Edara, (2007) Structural Optimization Simulation Studies in New Product Development, *SAE International*, 2007-01-4263.
5. Jim Zeich, Glen Peterson,(2003), Design Optimization Method for CAD Structural Components, *SAE International*, 2003-01-3427.
6. M.Borghi, F.Paltrinieri, (2005), Studying the Efficiency of a Compact Excavator Primary Workgroup Hydraulic Control System, *SAE International*, 2005-01-3618.
7. Jing Heng Wen, (2007), FEA Modeling Verification and Validation: Correlating Model with Test Data by Optimization Analysis, *SAE International*, 2007-01-1745.
8. Xinyu Ge, Jonathan Jackson, (2015), The Aritficial Intelligence Application Strategy in Powertrain and Machine Control, *SAE International*, 2015-01-2860.
9. Hyunkoo Kang, Wooyoung Jung, (2016), Modeling and Measurement of Payload Mass of the Wheel Loader in the Dynamic State based on Experimental Parameter Identification, *SAE International*, 2016-01-0469.
10. Nitin S Gokhale, Sanjay S Deshpande, Sanjeeve V Bedekar, Anand N Thite (2008), Practical Finite Element Analysis, *Finite to Infinite Institute*, 2008.
11. M.Surendran (2011), Stress Optimization of Rear Axle Control Arm,*initeCAE-Chassis Dept Benteler Engg Chennai Pvt. Ltd*, 2011.
12. Altair Engineering (2008), Numerical Methods in FEKO, *Altair Engineering 1820 Beaver Road Troy*, May 2008.
13. Zhifan Luo (2007), Benchmark of HyperStudy Optimization Algorithms, *Technical Specialist, Altair Engineering*, 2007.
14. Altair Engineering (2003), Light Weight SUV Frame, *Altair Engineering 1820 Beaver Road Troy*, May 2003.
15. Adrian Chapple, Dr. Adam Towse (2007), Delivering World Class Chassis Design, *ThyssenKrupp Automotive Tallent Chassis Ltd*, (2007).

