Proceedings of the International Conference on Manufacturing Systems – ICMaS
Vol. 4, 2009, ISSN 1842-3183
University POLITEHNICA of Bucharest, Machine and Manufacturing Systems Department
Bucharest, Romania

SAFETY AND RELIABILITY ASSESMENT OF FORKILFT TRUCK CABINE BASED ON VIRTUAL PROTOTYPE

Georgi TODOROV, Konstantin KAMBEROV, Borislav ROMANOV, Tsvetozar IVANOV

Abstract: *Virtual prototype model is used for safety and reliability assessment of a forklift truck cabin at the design stage. Numerical techniques are used to reproduce test conditions described in standard ISO 6055:2004 (E) and to find competitive solution based on cost and design properties criteria. Used approach accuracy is approved by physical tests carried out by certification organization. Application of virtual techniques at earliest possible product life cycle stage results in decreased costs for product implementation and decreased time-to-market.*

Key words: *reliability, virtual prototype, forklift, CAD, CAE.*

1. INTRODUCTION

Many regulatory programs and customer quality requirements have been the impetus for forklift trucks manufacturers to institute risk management processes utilizing both qualitative and quantitative risk assessment techniques. In some cases, the regulator or customer has prescribed the risk assessment techniques to be used for risk management, while in other cases there is leeway given to select a risk assessment technique of choice. Participation by environmental and safety compliance, operations, maintenance and engineering functions allows for risks to be properly ranked and for agreement on acceptable levels of residual risk [6].

Such regulations are stated in the ISO 6055: 2004 (E) standard, which contributes to forklift safety requirements. This standard is focused on overhead guards (OHG) for any type of high lift, rider operated, powered industrial truck with a lift height exceeding 1800mm. Any forklift should comply with the requirements and this is checked by specified in it test procedures. Generally, testing is the usual practice for the most of the forklift manufacturers, as it is required anyway. Thus, this verification and evaluation of design concept are possible after prototyping of the developed new product and any necessary changes will increase total development cost of the product and will increase time to market [3, 4].

An existing contemporary approach in mechanical design is the virtual prototyping (VP). It is giving the designers, testers and manufacturers a chance to reduce design errors and to optimise design and virtual testing in less time and in a risk-free environment. In the high dynamic industrial market VP has become a critically important way for a new design [2]. Generally, 80% of product reliability could be achieved in the design stage, saving expenses for physical prototypes and design modifications. Contemporary products development could be based on design-for-reliability approach, that has the benefits of performing earlier reliability assessment, at the stage of concept and design (Fig.1).

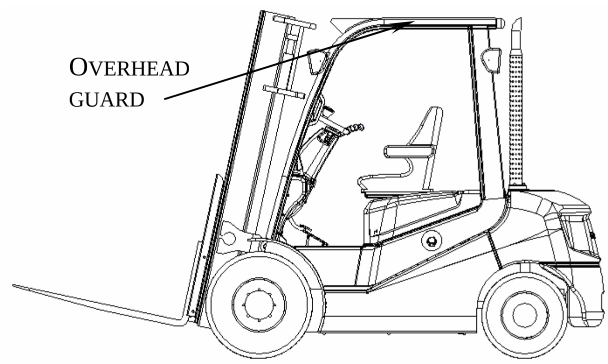

Fig. 1. Examined forklift component – Overhead Guard.

development, include target product operating life, improved probability and severity of failure, and reduced cost of support and maintenance [5, 7].

The application of VP approach to achieve product safety and reliability parameters enables to explore various design variants without prototyping and testing resources as time and money [1].

The goal of this study is to present practical case of virtual techniques, especially numerical methods, application to obtain cost effective, safe and reliable product – forklift truck. Examined different design variants allows to find cross point among design forms requirements, cost effectiveness and safety and reliability requirements. Final design variant is to be tested and verified by a physical prototype – as it is required by the certification organization. Both results – from virtual and physical prototype – are to be compared for validation of applied approach. All simulations over the built virtual prototypes will be performed using structural numerical simulations by Finite Element Method (FEM). This method has good practice and has been approved as standard tool for such type of simulations. Several software tools are available and evaluation of different designs are accessible for a large number of engineering bureaux and manufacturers. Presented engineering problem is typical for contemporary product development practice.

2. VIRTUAL PROTOTYPING AT DESIGN STAGE

Several OHG designs have been examined in order to determine optimal one. Their forms are based on aesthetic requirements and good visibility, combined with technology requirements, checked by comparative rigidity requirements.

2.1. Examined design variants

Four different designs of the overhead guard top plate have been examined and respectively, their virtual prototypes have been modelled. Their geometry and specifics are varies according to the above mentioned requirements. Comparison among them is performed using numerical techniques for simulation of sample load. Finite Element Analyses (FEA) are performed to obtain and explore force-deflection behaviour under this sample load. The results are compared and the best variant is to be elected for subsequent detailed design and analyses at ISO 6055: 2004 (E) requirements.

Two conflicting demands are available – wide visibility and sufficient rigidity. Each variant explores different possibilities for one or both characteristics. Another secondary, but important requirement, is to be technologic as form – i.e. not expensive. First variant is simple as design (technology) as has only one bend, the second one has more complex – and expected to be more rigid – shape, the third has increased visibility and complex shape and the last one has simple form, but has an additional side rib.

Proper finite element models have been developed based on the above stated variants. They are presented on Fig. 2 as mesh grids. Mesh density is selected to obtain as accurate results as are necessary for current research. This is supplied by built-in mesh check in the used software. Solid type elements are used instead of shell elements as to provide more detailed solution and accuracy. As the examined structures are symmetric, only a half has been modelled.

Initially, static steady-state load case under equally distributed sample load of total value 20 kN (top surface pressure applied) on the OHG plate has been examined. Applied load value is equal for all variants as to be used for comparison basis. Two types of constraints are applied – normal to middle plane for symmetry (constraint in normal direction only) and on the side of the OHG plate (vertical and longitudinal directions are constrained), where it is bolted or welded to the frame (depends on design variant).

Additionally, modal analysis is performed to determine first natural frequency. This parameter is important concerning possible vibrations. Computed modal shapes are indicative for design variant directional rigidity and for the total mass distribution of the structure. It uses the same boundary conditions as for the steady-state analyses, as are shown on Fig. 3, except the applied equally distributed load.

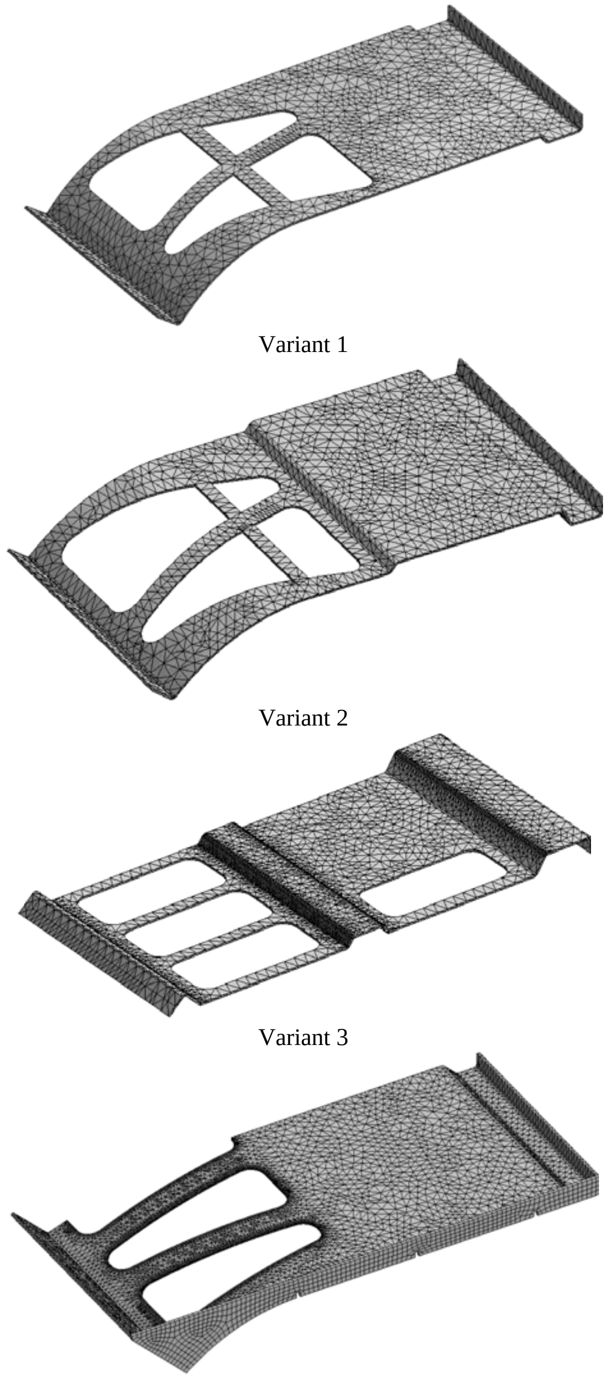

Fig. 2. Examined design variants.

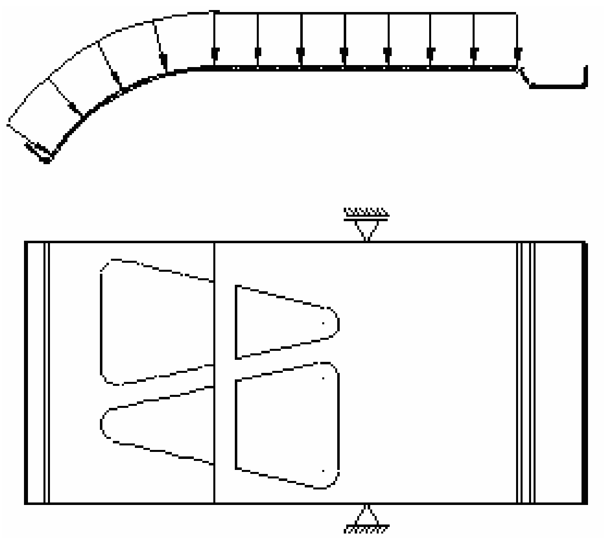

Fig. 3. Applied sample load and constraints.

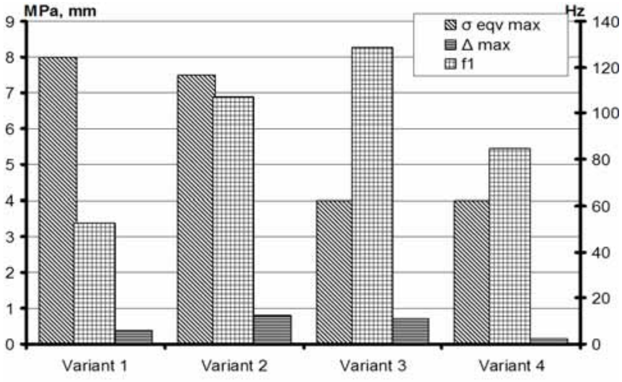

Fig. 4. Analyses results comparison.

The results from the performed analyses of different OHG top plate variants are compared as total deformation, equivalent (von Mises) stress maximal values and by first natural frequency on Fig. 4. Results show that *optimal behaviour has variant* 4 – its stress values are minimal and the first natural frequency is near to variants 2 and 3. Generally, it was expected result, as this variant has side welded plate, that has active role over its static and dynamic behaviour parameters. This variant is used in further examinations.

2.2. Test loads simulation for final design variant

The final design of the OHG – selected according to the previous examinations – is checked for conformance to ISO 6055: 2004 (E) standard requirements. It is important to note that the complete cabin is examined, including supporting tube frame. The next part of the truck – chassis – is relatively stronger and is not expected to have influence over the general truck behaviour. Thus, it is not included in the model.

Three simulations are performed in general:

- Lifting of entire truck by the OHG;
- Carried load falling over the overhead guard – two separate load cases according to the standard.

The simulations to be performed are steady-state – under proper loads for the both load cases marked above.

Thus, a simplified geometry of cabin only is to be examined, based on its virtual prototype model. The simplification consists of removing all unimportant for stiffness check parts as the engine cover for example. Some unimportant geometry features are removed too, as small chamfers, holes, etc. All welds are assumed to be of bonded type and no detailed study of their performance was carried out.

A FE model has been built, based on the above described geometry. Again, as the examined structure is symmetric by the longitudinal plane, only a half has been modelled.

Again, as for the previous analyses, a solid FE model is created. Contact elements are generated in common boundaries among structure components. All important and complex geometry bodies have higher density as they are expected to be of interest as stiffness behaviour. The meshed structure of entire model contains 152 000 nodes and 69 500 solid elements approximately. It is shown on Fig. 5.

Three separate load cases are examined, as are described above – when the whole truck is lifted by the overhead guard and the mast, and emergency cases: when the overhead guard is loaded by the lifted weight impact – in two variants. The applied boundary conditions are shown on the next Fig. 6.

Fig. 5. Examined FE model of final OHG design.

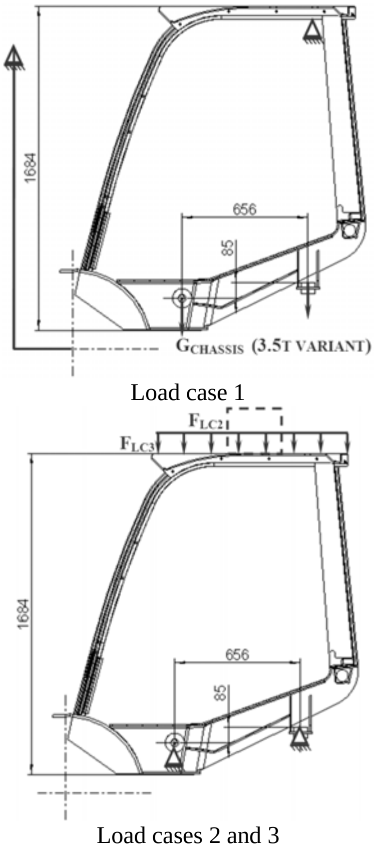

Fig. 6. Simulated load cases.

Load case 1 – lifting of the truck – has constraints on lifting points – at the front axle and at the top end side of the OHG plate. Load force for this load case is due to the gravity and mass distribution of the examined cabin.

The constraints for load cases 2 and 3 are applied on the connections to the chassis – placed at 656 mm distance. Input structural forces for these load cases are extracted from the standard ISO 6055: 2004 (E) – dynamic test (p.3.4.2 from the standard) and impact drop test (p.3.4.3 from the standard). The main target of these tests is to determine the resistance to permanent deflection of the portion of the OHG under which the operator sits or stands. All newly developed trucks are submitted to these tests.

Standard requirements for load case 2 are for dropping an oak object (300 mm × 300 mm impact area) of m = 45 kg mass of h = 1.5 m distance over the overhead guard, above the seat index point of the operator's seat in accordance with ISO 5353.

This corresponds to impact energy – 661.5 J, which is transformed in deformation energy. Above mentioned value for the needed impact energy is calculated as:

$$E_{imp} = m \cdot g \cdot h \,. \tag{1}$$

Corresponding forces, needed to apply in the FE models, are determined by iterations, starting with an initial simulation to determine deformation under certain load. An initial, sample, load is applied and the deformation is measured, then the initial impact energy could be calculated by the next formulae:

$$E_{ini} = F_{ini} \cdot \Delta_{ini} \,, \tag{2}$$

where F_{ini} and Δ_{ini} are initial force applied and corresponding to it deformation.

Thus, the calculated value is compared to required E_{imp} and, another force value is suggested, based on linear behaviour assumption. As the structure behaviour is not exactly linear, subsequent simulation could be needed to obtain exact matching of needed E_{imp} value.

Similar to the above described load case, this load case loads are determined for impact test energy (Etest in Table 1 in ISO 6055:2004(E)) of 21 760 J (truck rated capacity in interval 2 501 ÷ 3 500 kg). The impact force is determined in the same way as for the previous load case. Thus, the simulations are performed according to the above calculated force as steady-state analyses. The determined deformations and equivalent stress values will be evaluated as conformance to the standard requirements. These requirements are mainly geometric, where certain distance is required between seat and OHG, after the impact. Plastic deformations are allowable and, thus the main factor to evaluate the design is structural deformation.

The results from the performed analyses of the cabin assembly are shown on Figs. 7, 8 and 9 bellow as equivalent (von Mises) stress distribution field for each of the examined load cases. This is used to preview weak components and possible damages and to provide information for necessary design improvements. Maximal deformations of the OHG for each load case are as follows:

- Load case 1: 2mm;
- Load case 2: 17mm;
- Load case 3: 45mm.

Fig. 7. Equivalent stress distribution for load case 1.

Fig. 8. Equivalent stress distribution for load case 2.

Fig. 9. Equivalent stress distribution for load case 3.

First load case shows no critical stresses nor deformations. The second load case shows deformation in the limits – up to 20mm and the third load case does not decrease the distance to seat under 250mm – maximal deformations are about 45mm. Equivalent stress values for the first load case are minor, except some local stresses that are apparently due to mesh density or concentrators. Load case 2 shows equivalent (von Mises) stress value that shows elastic behaviour of the structure and no plastic deformations. Load case 3, the most critical, shows highest equivalent stress values that will definitively cause plastic deformations, but their values are still not destructive. Generally, neither deformations nor equivalent stresses calculated value are critical for the examined structure at stated loads.

Also, these results fully cover (as deformations) the required by the standard values. This allows proceeding further with this design variant. Only minor changes are applied, mainly connected to manufacturing technology, during design process.

3. PHYSICAL TESTING ON PROTOTYPE

All simulated on the virtual prototype and required by the standard tests have been performed on the stage of physical prototyping. These tests have been carried out by authorized company for ISO standard certification. It was used for certain design verifications too. Adjustable impact drop test load sample is used and it is shown on figure 10 below. It corresponds to used for the load case 3 simulations one and have been dropped from the required by the standard height that correspond to certain impact energy. Another test load is used for load case 2 – as it is described in chapter 2.2 above.

Fig. 10. Impact drop test sample load.

Fig. 11. Characteristic points for measurement.

Fig. 12. Impact drop test results.

Several points of interest (marked by "P") have been tracked on the prototype, as it is shown on Fig. 11 below. These points are characteristic and their deformation after impacts is to be measured and used for comparison to the simulations results and to check for standard conformity. Two separate cabins are tested for each of both dynamic and impact drop tests. All tests are performed over cabins, mounted on truck's chassis. This is not expected to influence the final results, as final plastic deformation is measured and no dynamic behaviour is explored. Deformations over different other parts – as dash board for example – is observed as to preview possible damages and to change design if it is needed.

The results from the tests are illustrated by a photo on Fig. 12, which corresponds to load case 3 – impact drop test. Measuring on the prototype deformation values after load test fully contributes to the simulation results. Excluding displacement, caused by bolted connection damage (marked on Fig. 12), the maximal deformation is about 35mm which is close to analysis results for deformation at load case 3 of 45 mm. All measured displacements for characteristic points, shown on Fig. 11, are relevant to simulation models results and are allowable.

Measured different result, compared to the virtual prototype, generally is caused by several factors, such as different material properties, model idealizations, manufacturing differences to the design, summary of the welding, etc. Another important factor, especially for the performed tests, is the damage of the bolted connection, which has changed the behaviour of the structure in general. Further design examinations for similar structures could include the bolted connections as it will increase model accuracy and will present more realistic structural behaviour. Asymmetric damage, as it appears to be, could be indicative for some mounting inaccuracies too.

Generally, the physical tests results shows sufficient coincidence to the virtual prototype analyses results.

4. CONCLUSIONS

Performed tests generally validate simulated tests and prove this approach as applicable for evaluating design solutions at earliest stages of product development. The product was certified directly and no corrective actions were needed over the design. This reflects on decreased expenses during product development at all and decreased time to market.

The following conclusions could be noticed, based on the above described performed simulations and physical tests:

- Safety and reliability assessment for forklift truck cabin is performed, implementing virtual prototyping technology, especially finite elements analysis /FEA technology;
- Performed design evaluation and optimisation at early product development stage, using large deflections simulations, gives opportunities for adequate prediction of drop test loads requirements;
- Physical prototyping and testing fully corresponds to simulation results, which is a validation of used technology for design evaluation through virtual prototyping;
- Adequate numerical simulation model have been developed and applied in industrial case for impact drop tests. It is applicable in similar case studies;
- A competitive product has been developed with decreased expenses for its development as iterative design solutions, released as physical models, are avoided.

REFERENCES

[1] Bertsche, B. (2008). *Reliability in Automotive and Mechanical Engineering*, Springer Verlag, Berlin.

[2] Connell, A. (2004). *The rise and rise of Virtual Prototyping*, Time-Compression Technologies, Vol. 12, Issue 3, (pp 44–49).

[3] *** *Industrial trucks – Overhead guards – Specification and testing*, ISO 6055: 2004.

[4] Todorov, G., Kamberov, K. (2008). *Risk Hazard Analysis of a Lifting Equipment with Articulate Kinematics*, International Conference Automatics & Informatics '08, Bulgaria, Sofia, October 1–4.

[5] Todorov, G., Todorov, N., Kamberov, K. (2001). *Reliability analysis Approach of high loaded power tool construction using Finite Element Analyses*, Proceeding of the Feature Modeling and Advanced Design-For-The-Life-Cycle Systems(FEATS), Valenciennes, France, (pp. 321–329).

[6] Trammell, S.R., Wright, R.D., Davis, B.J. (2001). *Using a Modified Hazop/FMEA Methodology for Assessing System Risk, Engineering Management for Applied Technology, EMAT 2001*, Proceedings of 2nd International Workshop, pp. 47–53.

[7] Tryon, R., Dey, A., Mahadevan, S. (2002). *Optimizing the design of mechanical components for reliability and cost*, International Journal of Materials & Product Technology, Vol. 17, Nos. 5/6, pp. 338–352.

Authors:

PhD, Georgi TODOROV, Proffessor , Technical University – Sofia, Faculty of Machine Technology,
E-mail: gdt@tu-sofia.bg

MSc, Konstantin KAMBEROV, Research Eng, Technical University – Sofia, Faculty of Machine Technology,
E-mail: kkamberov@3clab.com

MSc, Borislav ROMANOV, Research Eng, Technical University – Sofia, Faculty of Machine Technology, E-mail: bromanov@tu-sofia.bg

MSc, Tsvetozar IVANOV, Research Engineer, Technical University – Sofia, Faculty of Machine Technology,
E-mail: tsivanov@3clab.com