Journal of MECHANICAL ENGINEERING
– Strojnícky časopis

VOLUME 66, NO 2, 2016

Print ISSN 0039-2472
On-line ISSN 2450-5471

pp. 101 – 106
DOI:10.1515/scjme-2016-0024

MODELLING OF TWO-SEAT CONNECTION TO THE FRAME OF RAIL WAGON IN TERMS OF RESISTANCE AT IMPACT TEST

ČECH Rostislav[1], HORYL Petr[1,2], MARŠÁLEK Pavel[1,2]

[1] *VŠB – Technical University of Ostrava, Faculty of Mechanical Engineering, Department of Applied Mechanics, 17. listopadu 15, 708 33 Ostrava, Czech Republic, rostislav.cech.st@vsb.cz*
[2] *VŠB – Technical University of Ostrava, IT4Innovations, 17. listopadu 15, 708 33 Ostrava, Czech Republic*

Abstract: This contribution explores how design changes can affect resistance of rail wagon seat-to-frame connections at impact test and proposes alternative way to construct such connections. Results were due to a nature of presented problem obtained by computer modelling using MSC Nastran with explicit finite element method solution invoked. Physical tests were also conducted and comparison with numerical results is presented.

KEYWORDS: Welded connection, explicit fem, impact test, seat, rail wagon.

1 Introduction

Seats in trains are usually supported by steel footing that is connected to the frame of rail wagon by welded connections. This connection is where failure is the most likely to occur during impact and thus focus of this work. Because of that companies that produce these railway seats have to test them before homologation so that quality requirements are met. Tests presented in this work were done using custom built impact testing device (Fig. 2).

Fig. 1 Example of railway seat as produced by BORCAD cz s.r.o.

Footing of the seat is connected to the frame at the bottom and bumper is installed on the other side where seat usually is. Ram is dropped from a specific height with a specific weight to achieve desired amount of impact energy. Connections were tested with gradually increasing energy of the impact until failure to determine ultimate loading. First two design options (Fig. 3 and 4) were submitted to these tests.

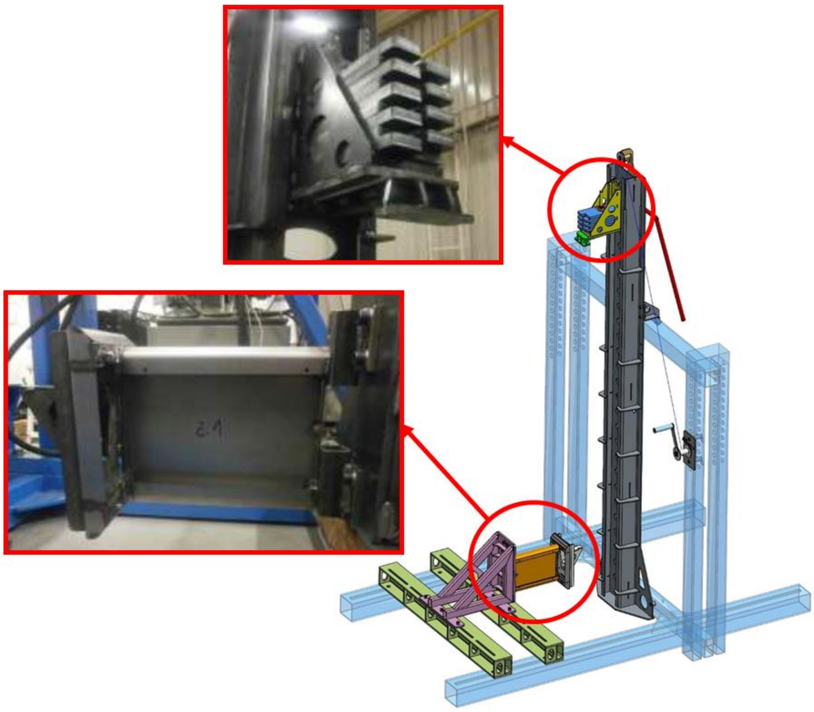

Fig. 2 Impact testing device constructed by BORCAD cz s.r.o.

First design option (Fig. 3) is simple cut sheet metal with circumferential welds. Because this connection failed to meet quality criteria second design option (Fig. 4) was proposed. Connector plate of second connection is cold-formed, circumferential welds are longer and few other short welds were introduced. For third design option (Fig. 5) different approach was suggested. Instead of relying on welds that usually cause problems with regard to consistent quality shape fit between two parts is preferred and welds are only secondary for load bearing. For detailed look on where welds and heat affected zones (HAZ) are situated see Fig. 7.

Because work was done in cooperation with company BORCAD cz s.r.o. and the company choose to withhold some internal information from public no specific values like material constants used for simulation can be presented further in the text.

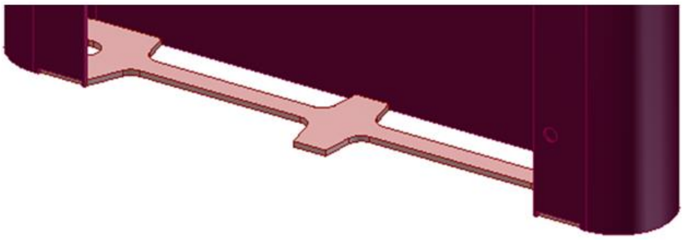

Fig. 3 Design option 1

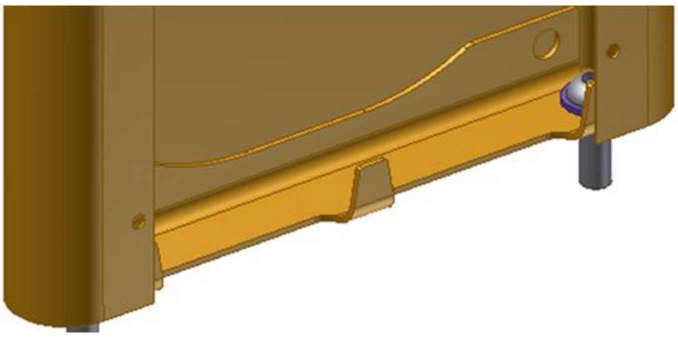

Fig. 4 Design option 2

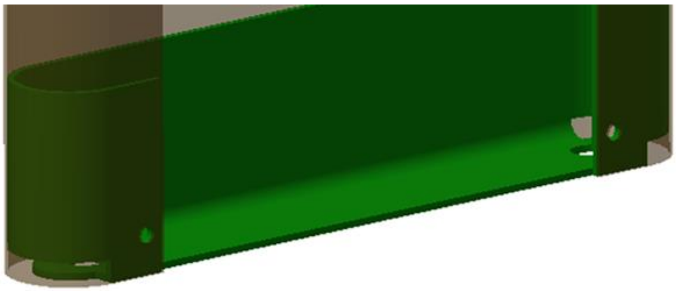

Fig. 5 Design option 3

FEM simulations of impact test were performed for all three design options.

2 Methodology

Quadrilateral shell membrane-bending elements were used to model sheet metal (CQUAD4). Majority of the model consist of elements with one integration point at plane and thus risks associated with zero-energy modes arise. Fully integrated elements were used only in contact regions. Three integration points were used through the thickness of the element.

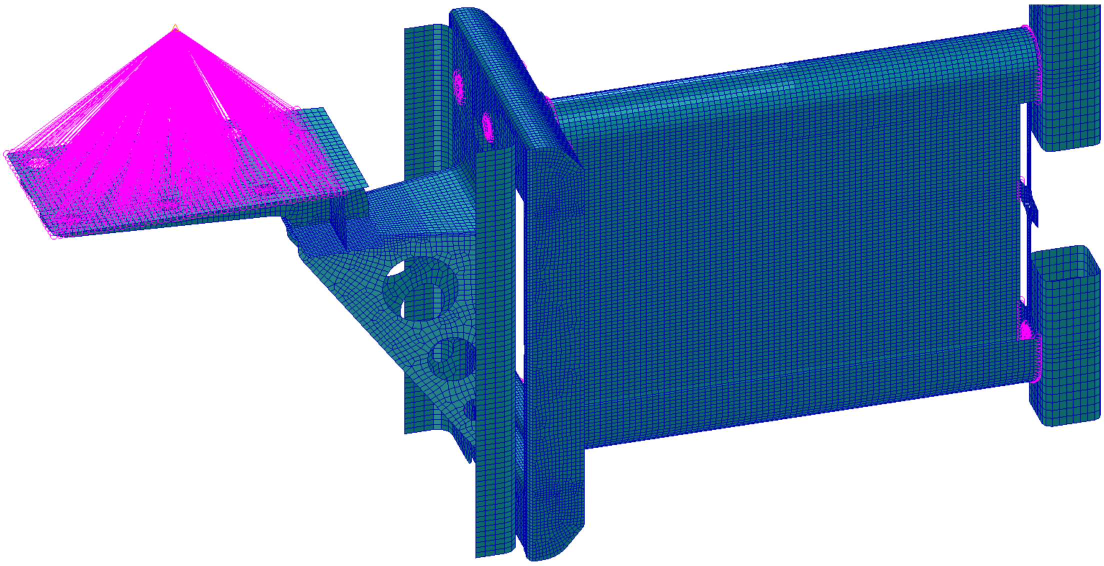

Fig. 6 Finite-element model

Only section of ram in contact is modeled and rest is replaced by rigid element (RBE2) connected to a mass point so that the ram has same mass as in physical tests (Fig. 6). All nodes of the ram have initial velocity. Both mass and initial velocity makes for impact energy of 566.8J (energy survived by both tested design options, see chapter 4). Bolted connections were simplified with beam (CBAR) and rigid element (RBE2). Welds are replaced with rigid elements (RBAR) and HAZ around them is modeled by using material with lower yield strength than that of the base material. Material model used to describe both the base material and the material in HAZ is simplified Johnson-Cook material model (without temperature dependency). Connection is done to steel section that is in reality welded to frame. The frame is absent in simulation and the steel section is instead fixed at the bottom.

Approach used to model welds is probably the easiest one and the most computationally efficient but certainly more precise methods could be used, for example replacing welds with beams of specific failure strength, modeling them with shell elements or even meshing whole welded area with solid elements for the most accurate but also the most computationally expensive solution. Since only comparison of different designs is important and absolute values not so much used method is found sufficient.

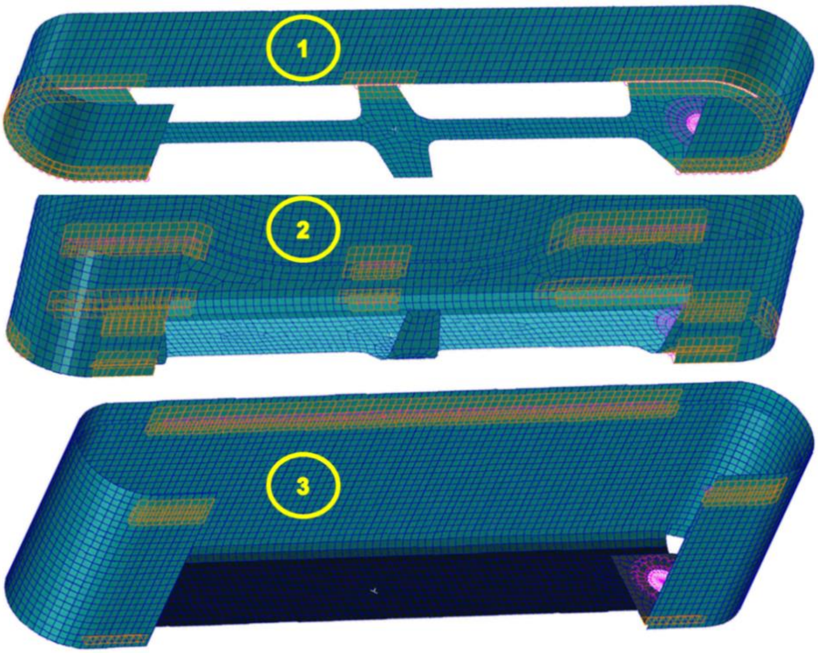

Fig. 7 Welds and heat affected zone for all three design options

All regions that are in a contact are modeled as frictionless as no exact coefficients of friction are known and this approach should give conservative results.

Explicit MSC.Nastran solver was used to model this test as significant plastic deformation occurs during very short period of time and speed dependent behavior of material is present.

Simulation ran until transient behavior diminished. That is for approximately 0.02s. Because no damping is used only elastic oscillation continues after this time indefinitely, that does not affect observed quantities. This approach is chosen because exact damping is not known and because this approach leads to conservative results.

3 Results

Results are not compromised due to so called hourglass energy since the hourglass energy levels during simulation did not exceed recommended 5% of internal energy. (Fig. 8)

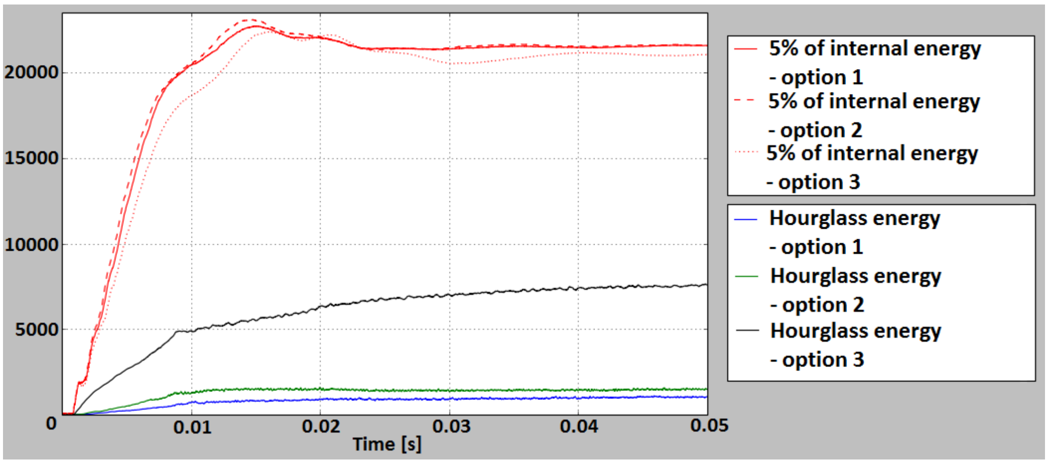

Fig. 8 Hourglass energy over time

Effective plastic deformation on elements in HAZ was used to evaluate risk of failure. Plastic deformation only grows during simulation and after transient behavior diminishes, remains constant. (Fig. 9) Layer of shell (integration point through thickness) displayed is the one with the highest values. Critical value for used material is displayed in dotted line.

Effective plastic deformation is further on presented as a fraction of the critical value for given material.

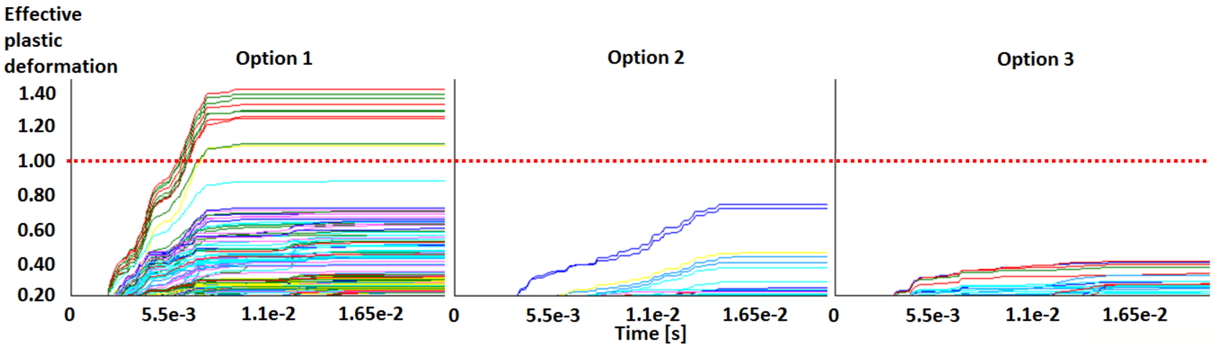

Fig. 9 Effective plastic deformation of heat affected zone over time

Maximum values achieved during simulation are summed in Tab. 1. It is clear that based on maximum value over time the first design option is the worst option. Critical value is exceeded, risk of failure is present. Maximum effective plastic deformation on second option is significantly lower, safely under critical value and there is no reason to believe that this connection would not pass tests. Third option is even better that the second. From plots on Fig. 9 it is also clear that the first design option undergone the highest amount of damage with second and third design option having by far better results.

Tab. 1 Maximum effective plastic deformation

	Design option 1	Design option 2	Design option 3
$\varepsilon_{ef,\ max.}$ [-]	1.400	0.748	0.398

2 Comparison with experimental data

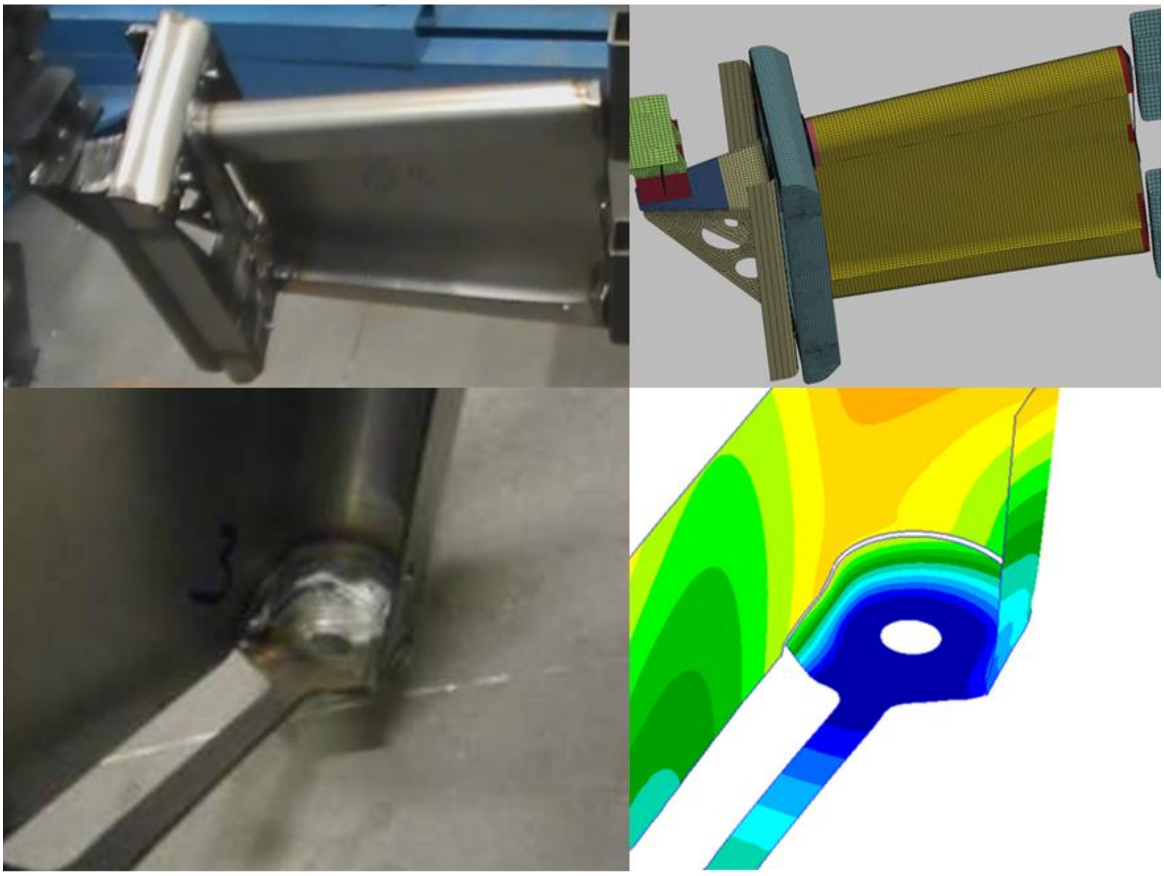

Fig. 10 Deformation comparison

Maximum impact energy that connections did withstand during physical tests on impact device is summed in Tab. 2. It can be observed that the second design option was truly more durable by about 44.4%.

Tab. 2 Maximum impact energy

	Design option 1	Design option 2
$E_{max.}$ [J]	566.8	818.2

Because there is not enough information about how would welded connection really behave after reaching critical value only maximum achieved values during simulation can be compared. (Tab. 1) It shows that second design option would be about 87.2% more durable that the first design option.

From qualitative point of view there is good correlation of results. The second design option was proven to be more durable both by experiment and simulation.

Limited static comparison is provided (Fig. 10) that shows similar plastic deformation after dynamic loading diminished overall and even in the connection detail.

CONCLUSION

Both experimental testing and numerical simulations showed that durability of connection can be improved by increase in welded loadbearing surface and good correlation between both approaches was achieved as similar difference in durability and same manner of deformation while undergoing dynamic loading was observed.

Other design option was then proposed that introduced shaped fit instead of increasing volume of welding used and compared to others with validated computer model. Results suggested that it is safer option as two loadbearing mechanism are introduced rather than one and numerical results outperformed by far previous designs.

Overall work showed that computer modelling techniques can give reliable results for impact loading and thus can be used to lower costs of development as unlimited amount of design option can be assessed prior to physical testing with little or no effort once computer model is done and only geometric details of the connection are changed.

ACKNOWLEDGEMENT

The work was supported by VŠB – Technical University of Ostrava under grant SGS SP2016/176 and project „IT4Innovations excellence in science - LQ1602“ supported by The Ministry of Education, Youth and Sports from the National Programme of Sustainability. (NPU II)

REFERENCES

[1] Čech, R.: Modelling Two-seat Connection to the Frame of Rail Wagon in Terms of Resistance at Impact Test. Ostrava: VŠB – TUO, 2015. Diploma thesis. (in Czech)

[2] Masopust, J.: Impact Resistance Test of Welded Two-seat Connection. Fryčovice: BORCAD cz s.r.o., 2014. Technical report. (in Czech)

[3] Wu, S. R., Gu, L.: Introduction to the Explicit Finite Element Method for Nonlinear Transient Dynamics, New Jersey, John Wiley & Sons, Inc., 2012. ISBN 9780470572375.

[4] Abdollahpoor, A., Marzbanrad, J.: Crashworthiness study of axial impact in cylindrical aluminium tubes. *Journal of Mechanical Engineering – Strojnícky časopis*, 2010, Vol. 61, 1, (p. 1-18), ISSN 0039-2472.