The 40th International Conference on
Mechanics of Solids, Acoustics and Vibrations &
The 6th International Conference on
"Advanced Composite Materials Engineering"
COMAT2016 & ICMSAV2016
Brasov, ROMANIA, 24-25 November 2016

THEORETICAL STUDIES USED AT A TOWING ASSEMBLY. PART I

Andrei Victor Petrici[1], N. Gheorghe Radu[2], Ioana Comanescu[3]
[1] TRANSILVANIA University, Brasov, ROUMANIA, petriciandreivictor@yahoo.com
[2] TRANSILVANIA University, Brasov, ROUMANIA, rngh@unitbv.ro

Abstract: *This paper contains theoretical studies with Finite Element Analyses for a complex coupling consisting of a towbar (for Audi Q5), a transverse frame to grip on vehicle chassis and the necessary elements for fitting (flanges, screw-nut system, welding).*
It was proceeded first with the conception and design of all components but also the assembly and fastening system.
For the execution and analyze of the model, it was started by shaping it, meshing it with shell and solid type of elements and analysis of the state of tension and deformation under load.
Towing assembly contains also fastening elements by welding.
Keywords: *towbar, displacement, load, stress*

1. INTRODUCTION

The current paper contains a lot of studies with Finite Element Analyses for a complex towing assembly used at an Audi Q5 vehicle, figure 1 and figure 2. It was considered that the car is in braking time and it has a trailer without brake which presses the whole towing assembly.
From the beginning, we should know that these results on this paper will be compared with some results from another paper which has experimental determinations on a similar towing assembly.
In other studies, it has been shown the higher brake force introduces the higher stresses in the towing assembly. If the car with trailer have a frontal crash, then appear the greater stresses in the towing assembly, and the speed is higher so the stresses are higher.

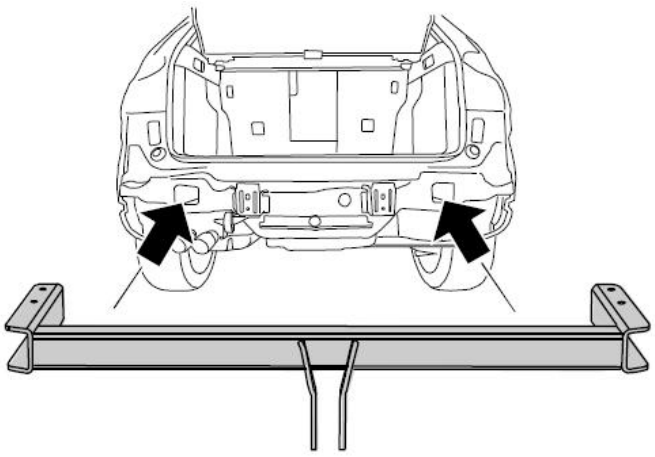

Figure 1: Audi Q5 – towing assembly

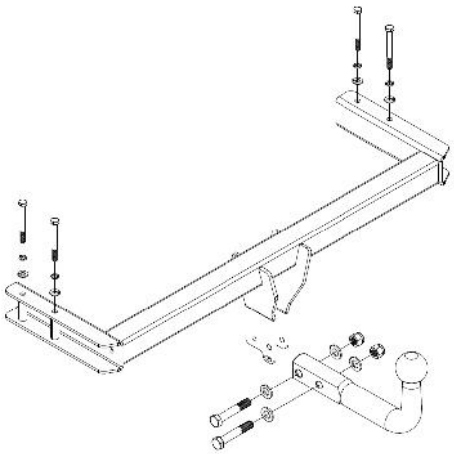

Figure 2: Towing components

The towing assembly is assembled on the vehicle with 4 screws M10x100, and it contains two pairs of flanges and a square pipe in the middle, figure 1 and figure 2. The all 4 flanges are welded on the square pipe and the

towball is fixed with two M12x70 screws and two M12 nuts. All components are from S355J2 material. Theoretical studies comply exactly as catching real constraints on vehicle.

2. FINITE ELEMENT ANALYSIS

This analysis was done with the same conditions that the towing assembly is working on the vehicle's chassis. The towing assembly was considered fix in the 4 M10x100 screws and the load pushed in the center of the towbar's ball. The loads were similar, 7,5 [kN], 15 [kN], 25 [kN] and 50 [kN]; there was taken these values because it was considered different braking and crashes.
For this analysis, we did not use cycles, because the software is not capable to do this type of test without the material curve, and it worked only in elastic region, without residual stress.
To simplify the analyzed model, the assembly was considered without ball, it had only one point. It could do this thing because the highest stress is in the curvature radius, not in the ball, figure3.

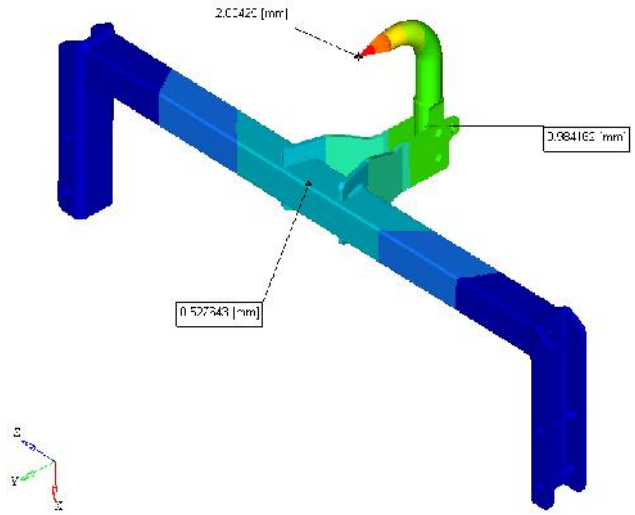

Figure 3: Area with maximum strain

For the maximum strain, there is attached a table (Table 1), where can be seen the displacements for each load. In Inventor software, the sample was measured (without deformations) and in HyperMesh software the displacements were measured with the allocated load, in all the 3 points (see fig. 3).

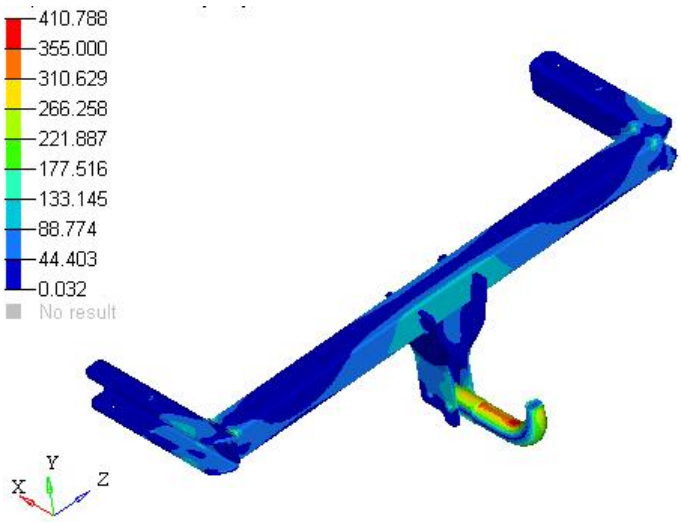

Figure 4: Area with maximum stress

It can be seen in figure 4 that the maximum stress is in towbar, so this component is the most requested from the whole assembly.
For this FEA the mesh was with elements solid type and they were used 324263 elements and 389167 nodes.

In Table 2 the results are attached and they will be compared later.
It was taken these three points for measurement because in the experimental determination we will can measure also these points with a vertical caliper and we will can make comparisons between the results.

Table 1: Strains for the all three points for all the loads

Load [kN]	Point	Before test [mm]	Strains [mm]	After test (with load) [mm]
7,5	1	520,92	2,03	522,95
	2	312,50	0,98	313,48
	3	311,00	0,53	311,53
15	1	520,92	4,06	524,98
	2	312,50	1,96	314,48
	3	311,00	1,06	312,06
25	1	520,92	6,77	527,69
	2	312,50	3,27	315,77
	3	311,00	1,77	312,77
50	1	520,92	13,54	534,46
	2	312,50	6,54	319,04
	3	311,00	3,54	314,54

3. CONCLUSION

Comparing results FEA and experimental determination (cycles), the main conclusion is the areas where the maximum stress appear are similar, excepting the towbar. Pink color means that in those regions are micro cracks and there were the highest stresses.

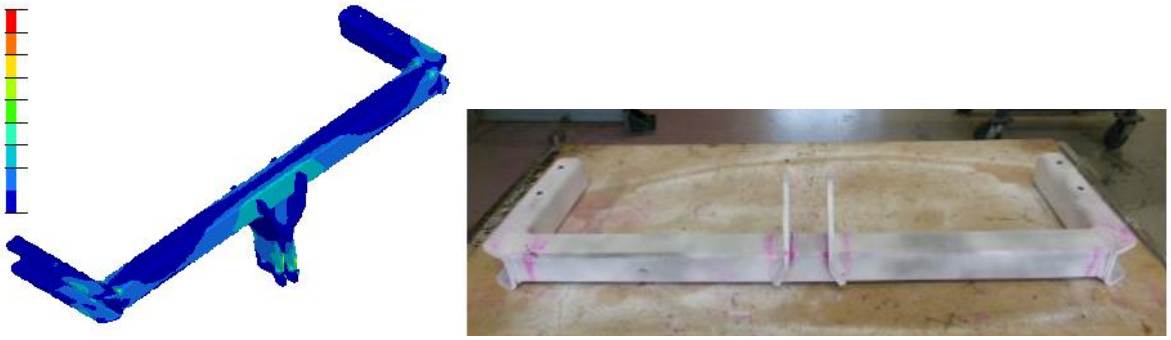

Figure 5: Comparison between the testing methods

Referring to the towbar, in the past the same authors were shown that the best section for this component is with trapezoidal section, but this thing complicates the manufacturing process, figure 5.

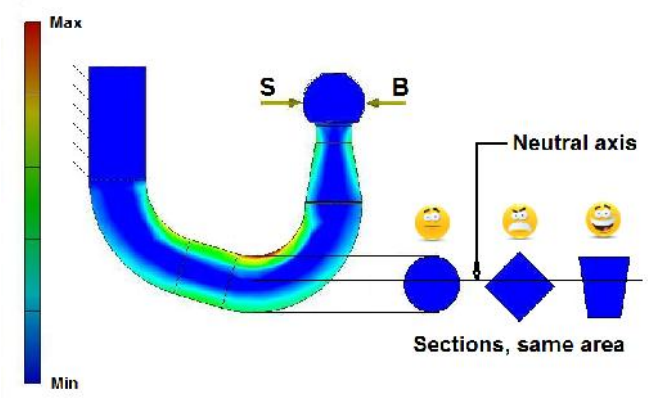

Figure 6: Different sections for towball

REFERENCES

[1] Ponomariov, S. D., s.a, Calculul de rezistenta in constructia de masini, vol.1, vol. 2, Editura Tehnica, Bucuresti, 1960.
[2] Untaru, M., Campian, V., Ionescu E., Peres, Gh., Ciolan, Gh., Todor, I., Filip, N., Campian, O., Dinamica Autovehiculelor, Reprografia Universitatii Transilvania din Brasov, Brasov, 1989.
[3] Radu, N. Gh., Rezistenta materialelor si elemente de teoria elasticitatii, vol. 1, Editura Universitatii "Transilvania" din Brasov, Brasov, 2002.
[4] Radu, N. Gh., Rezistenta materialelor si elemente de teoria elasticitatii, vol. 2, Editura Universitatii "Transilvania" din Brasov, Brasov, 2002.
[5] Radu, N. Gh., Comanescu, Ioana, Rosca, I. C., Elasticitate, plasticitate, rupere – teorie si aplicatii, Editura Universitatii "Transilvania" din Brasov, Brasov, 2009.
[6] Stancescu, C., Modelare parametrica si adaptiva cu Inventor, Editura Fast, Bucuresti, 2009.
[7] Stancescu, C., Manolache, D. S., Parvu, C., Ghionea, I. G., Tarba, C., Proiectare Asistata cu Autodesk Inventor – Indrumar de laborator – Editia a II-a, Editura Fast, Bucuresti, 2012.
[8] HyperMesh user guide