

¹Gheorghe N. RADU, ²Calin ITU, ³Andrei Victor PETRICI

THEORETICAL STUDIES (FEA) AND EXPERIMENTAL DETERMINATIONS USED AT A TOWING ASSEMBLY

¹⁻³. Transilvania University, Brasov, ROMANIA

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; it will simulate and determine experimentally the behavior of the towing assembly in compression conditions (braking). The sample contains fastening elements by welding. Finally, it was done a comparison of theoretical and experimental results; further work will follow on some optimization of the components from the studied assembly.
Keywords: towbar, displacement, load, stress

INTRODUCTION

This paper contains dynamic tests at a towing assembly used at Audi Q5 vehicle, fig. 1-1. The experimental determinations were made by a MTS traction machine; the models are made in Inventor and Finite Element Analyses with HyperMesh software.

Additional there will be done some comparisons between a dynamic test at 2×10^6 cycles and FEA on the regions where the maximum stresses appear.

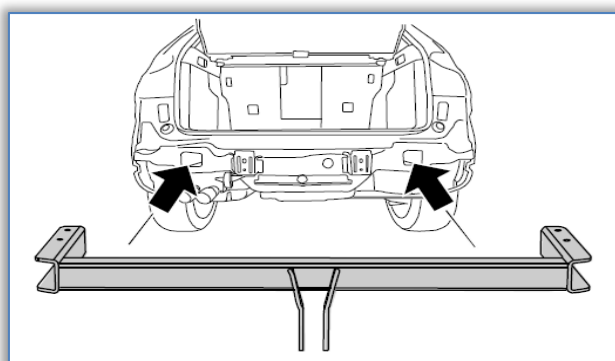


Figure 1-1: Audi Q5 – Towing assembly

The towing assembly is assembled on the vehicle with 4 (four) screws M10x100, and it contains two pairs of flanges and a square pipe in the middle, fig. 1-2. 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.

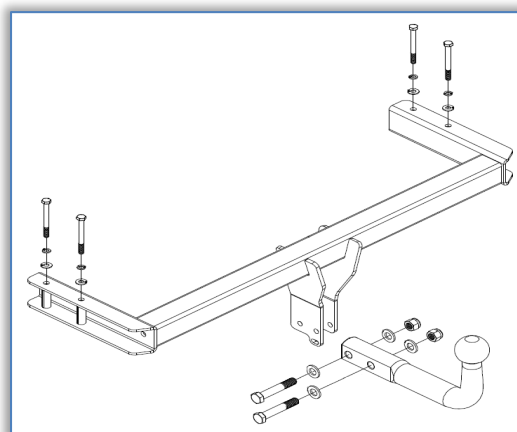


Figure 1-2: Towing components

EXPERIMENTAL DETERMINATIONS

In general, this type of towing assembly is tested with an approximate 7,5 [kN] load at 2×10^6 cycles, on all three axes, the name of the test is Carlos. In picture no. 2-2 is appeared a sample from this test and there they can see the areas where appear cracks after the 2×10^6 cycles, at a dynamic load. The pink color represents regions where the maximum stresses appear.

In our tests we tested a little bit differently. It simulated only the moment when the car is braking and the trail pushes the car entering stress on the towing assembly. The loads were: 7,5 [kN] – 20 cycles, 15 [kN] – 10 cycles, 25 [kN] – 10 cycles and 50 [kN] – 5 cycles.



Figure 2-1: Towing assembly on the test bench

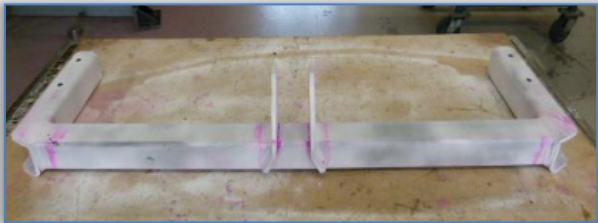


Figure 2-2: Towing components

For the experimental determinations, there were used some devices to adapt this type of project at the traction machine. There was made a rigid base plate with 20 [mm] thickness, fig. 3-1, and two other devices to fix the towing assembly, fig. 3-2.

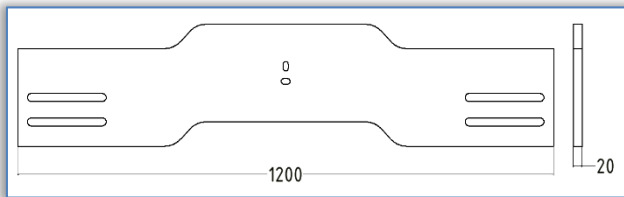


Figure 3-1: Rigid base plate

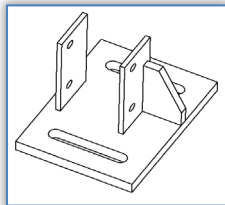


Figure 3-2: Left – Right devices

Most of the parts of the devices are made by a cutting laser machine and after that they are welded to form a new component, fig. 3-2. The sample is fixed with 2 devices (fig. 3-2), then the two devices will be fixed on the rigid base plate (fig. 3-1) and finally, all assembly will be fixed on the traction machine, fig. 4-1.

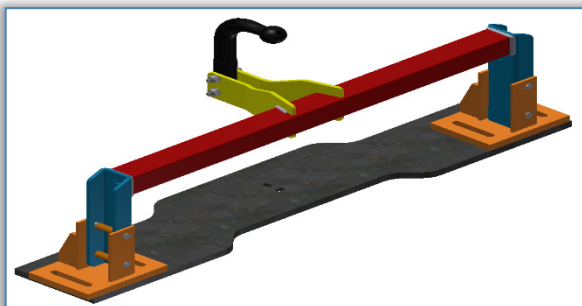


Figure 4-1: Assembly for traction machine



Figure 4-2: MTS traction machine

The assembly from figure 4-1 will be fixed on the support of traction machine with 2 screws. The rigid base plate has 2 holes to be fixed on the traction machine support.

The traction machine is from MTS (fig. 4-2) and it has 1.000 [kN] maximum capacity. This machine was adapted to do cycles and to give diagrams. All the assembly was tested at the all loads until it was destroyed. The towing assembly was not broken, but the material was in plastic region. For the 7,5, 15 and 25 [kN] load the assembly worked only in elastic region, but it remained residual stress field.

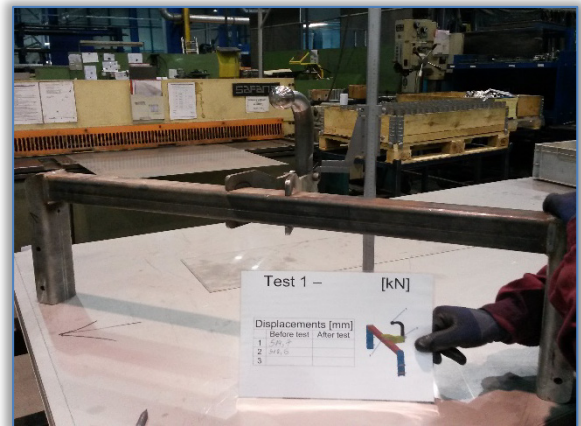


Figure 5-1: Vertical caliper measurement

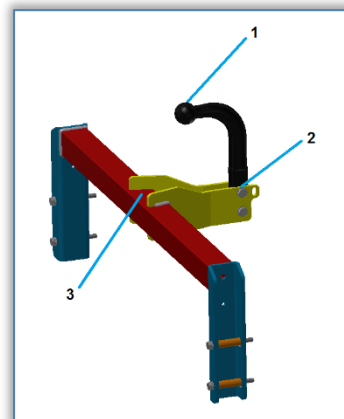


Figure 5-2: The 3 points measured

Before and after the tests the towing assembly was measured with a vertical caliper (fig. 5-1) in 3 points (fig. 5-2), to see the deformations resulting from residual stress. The results are in Table 1, where it can be seen the measurement before and after the 4 tests and in figure 6-1 it can see how the towing assembly is fixed on the traction machine. For the last column from Table 1, Ball displacement, the values were read from diagrams.

Table 1: Experimental determinations values

| Load [kN] | Point | Before test [mm] | After test [mm] | Measured deformation [mm] | Ball displacement [mm] |
|-----------------|-------|------------------|-----------------|---------------------------|------------------------|
| 7,5 – 20 cycles | 1 | 514,70 | 512,76 | 1,94 | 5,43 |
| | 2 | 312,60 | 311,36 | 1,24 | - |
| | 3 | 310,62 | 310,78 | -0,16 | - |
| 15 – 10 cycles | 1 | 512,76 | 511,84 | 0,92 | 8,37 |
| | 2 | 311,36 | 311,16 | 0,20 | - |
| | 3 | 310,78 | 311,70 | -0,92 | - |
| 25 – 10 cycles | 1 | 511,84 | 511,00 | 0,84 | 14,06 |
| | 2 | 311,16 | 310,82 | 0,34 | - |
| | 3 | 311,70 | 310,74 | 0,96 | - |
| 50 – 5 cycles | 1 | 511,00 | 497,00 | 14 | 35,00 |
| | 2 | 310,82 | 308,64 | 2,18 | - |
| | 3 | 310,74 | 309,32 | 1,42 | - |

For the last test (at 50 [kN]) the towing assembly could not touch 50 [kN] because it was in plastic region, the maximum load was 42 [kN]. After this load the material started to flow, the strain was very high and also there was only one cycle.



Figure 6-1: Towing assembly fixed on traction machine support

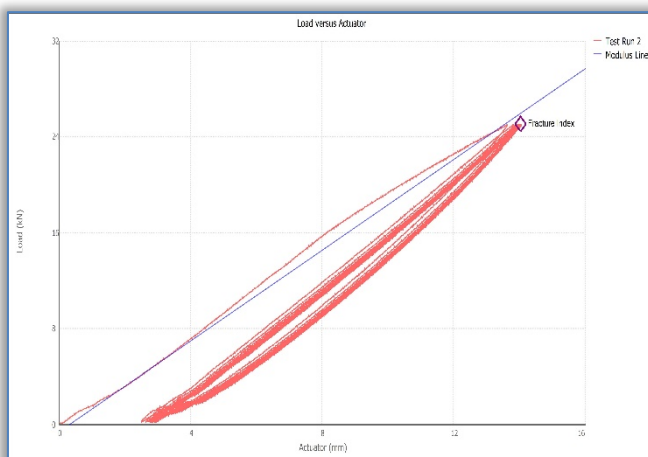


Figure 6-2: Load – Displacement diagram

In figure 6-2 is attached a diagram from traction machine with 25 [kN] load and 10 cycles. For all the tests the traction machine gives this type of diagrams. On diagrams it can read the maximum load and the maximum displacement. The dates for Table 1 are from these diagrams. Only for 50 [kN] test the diagram appears without cycles, it shows like a normal diagram load – strain or stress – strain.

For simple traction the machine can make the stress and strain diagram, if it is known the section area.

In Table 1 are 2 negative values because in the square pipe appears a torsion strain, higher than bending strain and the edge of the pipe is moving up. After a higher load the pipe starts to bend and the deformations are positive.

After the last test, when the load was the highest and material was in plastic region, it was discovered that the most deformed part of the assembly was the towbar, and only the towbar was in plastic region. In the past, the same authors demonstrated the strongest section for the towbar is with trapezoidal section and the weakest is with rhombic section. In these experiments there was used a towbar with round section which is between the two others, but it is the cheapest solution, fig. 7.

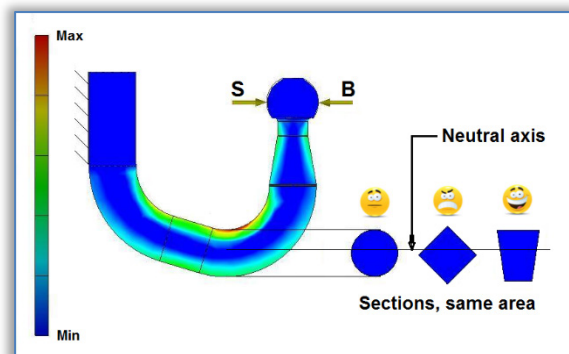


Figure 7: Different sections for towbar

Maybe if we used a towbar with trapezoidal section, the assembly could have been tested at a higher load.

FINITE ELEMENT ANALYSIS

This analysis was done with the same conditions and loads as the experimental determinations. 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].

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, fig. 8-2.

For this FEA the mesh was with elements solid type and they were used 324263 elements and 389167 nodes.

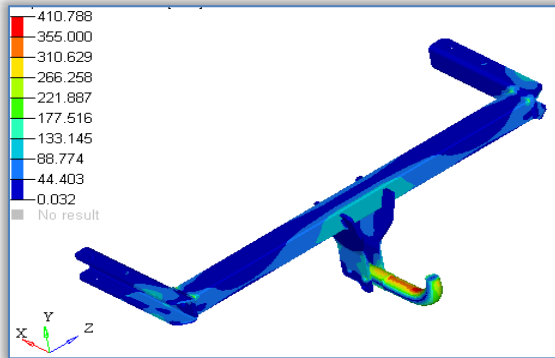


Figure 8-1: Area with maximum stress

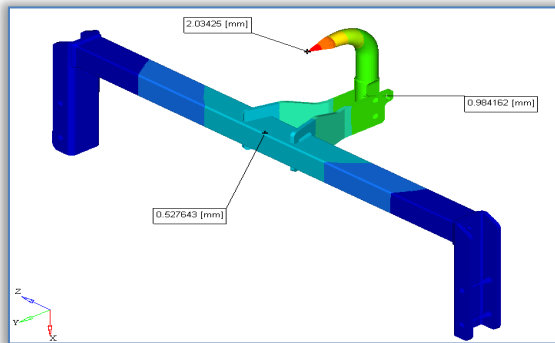


Figure 8-2: Area with maximum strain

For the maximum strain there is attached a table, 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. 8-2). In Table 2 the results are attached and they will be compared later.

Table 2: FEA values

| 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 |

It can be seen in the Table 3 the displacements are linear, but in experimental determination the sample's material is in plastic region and the strains higher.

The experimental strains are higher (approximate double) than virtual strains. In the future is interesting to do some classical calculations to determine why these high differences exist and to use strain gauges. Also in the future the authors want to develop a similar system, but with composite

elements, to do the sample lower weight, because the car makers tendency is to do the auto vehicles increasingly lighter.

Table 3: Comparison experimental determinations & FEA

| Load [kN] | Point | Experimental Strains [mm] | Virtual Strains [mm] | After test (with load) [mm] |
|-----------|-------|---------------------------|----------------------|-----------------------------|
| 7,5 | 1 | 5,43 | 2,03 | 522,95 |
| 15 | 1 | 8,37 | 4,06 | 524,98 |
| 25 | 1 | 14,06 | 6,77 | 527,69 |
| 50 | 1 | 35,00 | 13,54 | 534,46 |

CONCLUSION

Comparing the pictures no. 2-2 and no. 8-1 they can see some similarities. The maximum stresses appear in the same areas, except only the towbar. In Carlos's test appeared stresses (micro crakes) only near welding regions.

Note

This paper is based on the paper presented at The 1st International Conference "Experimental Mechanics in Engineering" - EMECH 2016, organized by Romanian Academy of Technical Sciences, Transilvania University of Brasov and Romanian Society of Theoretical and Applied Mechanics, in Brasov, ROMANIA, between 8 - 9 June 2016

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., 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.

copyright ©

University POLITEHNICA Timisoara,
Faculty of Engineering Hunedoara,
5, Revolutiei, 331128, Hunedoara, ROMANIA
<http://acta.fih.upt.ro>