ANALIZA STATICĂ A INTERACŢIUNII ROATĂ-ŞINĂ PRIN FOLOSIREA PROGRAMULUI ANSYS

A STATIC WHEEL-RAIL INTERACTION ANALYSIS BY USING ANSYS PROGRAM

Prof. Eng., PhD. Ioan SEBEŞAN[1], Eng. Yahia ZAKARIA[2]

[1]Faculty of Transports, "Politehnica" University of Bucharest
e-mail: ioan_sebesan@yahoo.com
[2]*e-mail: sharkasianlion@yahoo.com*

Rezumat: Un proces de simulare completă aplicată pe osia montată a locomotivei Deisel-electrice este descris în această lucrare. Tensiunea, deformarea şi starea de contact sunt obiectivele acestui studiu. La sfârşit, rezultatele procesului de simulare sunt verificate pentru a se asigura că acestea sunt aproape de rezultatele reale.
Cuvinte cheie: Osia montată, Ansys, analiza, interacţiune, simulare.

Abstract: A full simulation process applied on a locomotive wheelset is described in this paper. Stresses, deformations and contact status are the targets of this study. At the end, results of the simulation process are verified to ensure that they are close to real results.
Keywords: wheelset, Ansys, analysis, interaction, simulation.

1. INTRODUCTION

The interaction between wheel and rail has an important role in dynamic and tractive performances determination on railway vehicles. The tractive force created on locomotives' drive wheel surfaces is widely related to the contact zone characteristics such as: wheel-rail contact zone dimensions, friction coefficient between wheel and rail, etc. The more accurate researches about the interaction between wheel and rail are made, the more illuminated understanding of phenomena related to traction process is gained.

In this work paper, the wheel set used on C.F.R. 060-DA Romanian Diesel-electrical locomotives is studied. Steps of simulation process, which are described in [1], have been applied on this wheel set using Ansys simulation program.

Every simulation contains the following stages (fig2):

1- Preprocessing: - material characteristics definition, - creating geometric model, - meshing, - applying restrictions and loads. In this stage, types of analyzing should be chosen: static, modal, thermal, etc. Decisions about considered linkages between the components and considered contact between parts options are also here taken, i.e. linear or iterative, etc.

2- Solution.

3- Post processing and results viewing.

4- Results verification.

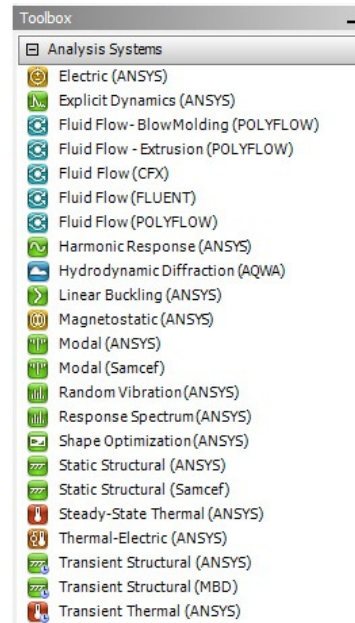**Fig.1 Workbench simulations**
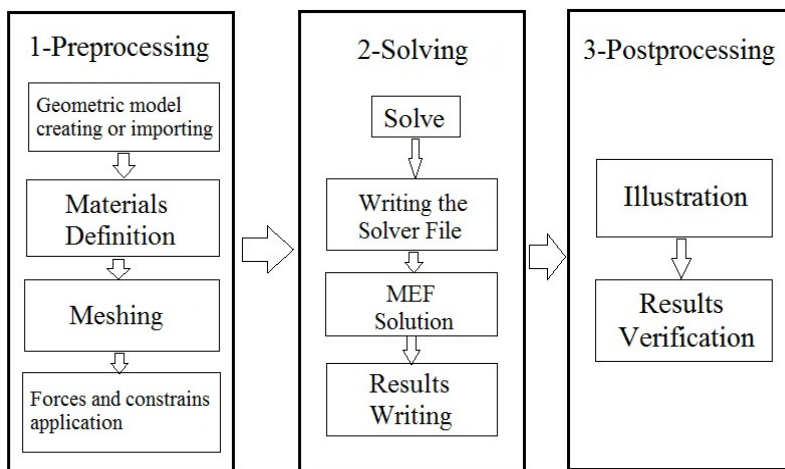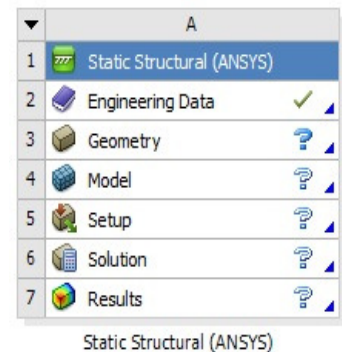
Fig.2 Simulation stages.

Fig.3 Static analysis - ANSYS

2. MATERIALS DEFINITION (ENGINEERING DATA)

Characteristics of chosen materials for studied parts are shown in fig.4. For the materials which are not included in Ansys library, it is recommended to visit the website [5], which has a larger library with materials classified and introduced according to the symbols used in different

countries. The axles of C.F.R. 060-DA Diesel-electrical locomotives are made of steel DC 35, while wheel bandages are made of steel DC 60, having shear resistance in traction mode $\sigma_r = 75 \dots 90 \ kgf/mm^2$.

Fig.4 Material characteristics.

Fig.5 Density of some metals.

Young Modulus of Elasticity - *E* - (10⁶ psi)														
Metal	Temperature (°C)													
	-200	-129	-73	21	93	149	204	260	316	371	427	482	538	593	649
	Temperature (°F)													
	-325	-200	-100	70	200	300	400	500	600	700	800	900	1000	1100	1200
Cast iron														
Gray cast iron				13.4	13.2	12.9	12.6	12.2	11.7	11.0	10.2				
Steel														
Carbon steel C <= 0.3%	31.4	30.8	30.2	29.5	28.8	28.3	27.7	27.3	26.7	25.5	24.2	22.4	20.4	18.0	
Carbon steel C => 0.3%	31.2	30.6	30.0	29.3	28.6	28.1	27.5	27.1	26.5	25.3	24.0	22.2	20.2	17.9	15.4
Carbon-moly steels	31.1	30.5	29.9	29.2	28.5	28.0	27.4	27.0	26.4	25.3	23.9	22.2	20.1	17.8	15.3
Nickel steels Ni 2% - 9%	29.6	29.1	28.5	27.8	27.1	26.7	26.1	25.7	25.2	24.6	23.0				
Cr-Mo steels Cr 1/2% - 2%	31.6	31.0	30.4	29.7	29.0	28.5	27.9	27.5	26.9	26.3	25.5	24.8	23.9	23.0	21.8
Cr-Mo steels Cr 2 1/4% - 3%	32.6	32.0	31.4	30.6	29.8	29.4	28.8	28.3	27.7	27.1	26.3	25.6	24.6	23.7	22.5
Cr-Mo steels Cr 5% - 9%	32.9	32.3	31.7	30.9	30.1	29.7	29.0	28.6	28.0	27.3	26.1	24.7	22.7	20.4	18.2
Chromium steels Cr 12%, 17%, 27%	31.2	30.7	30.1	29.2	28.5	27.9	27.3	26.7	26.1	25.6	24.7	23.2	21.5	19.1	16.6
Austenitic steels (TP304, 310, 316, 321, 347)	30.3	29.7	29.1	28.3	27.6	27.0	26.5	25.8	25.3	24.8	24.1	23.5	22.8	22.1	21.2

Fig.6 Modulus of elasticity (Young)

[5]

3. MECHANICAL MODEL (MECHANICAL)

The mechanical model is the base where all other analysis stages start from, and it can be built or constructed using Ansys itself or by using another mechanical drawing program, as it was followed in this paper, where mechanical model is made completely in Solidworks program then imported into ANSYS as solid bodies having four contact zones, two of them are between the axle and the wheels and two are between wheels and rails. Shapes and dimensions of axles, wheel centers and wheel bandages are respectively shown in fig.7, fig.8 and fig.9.
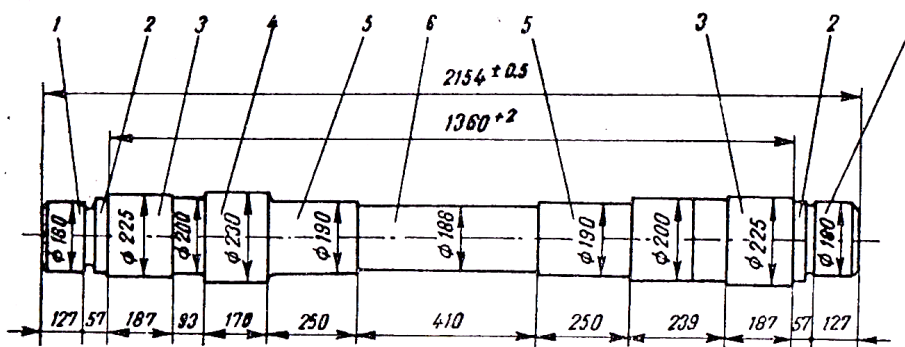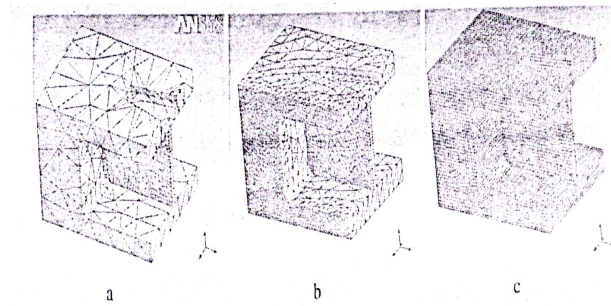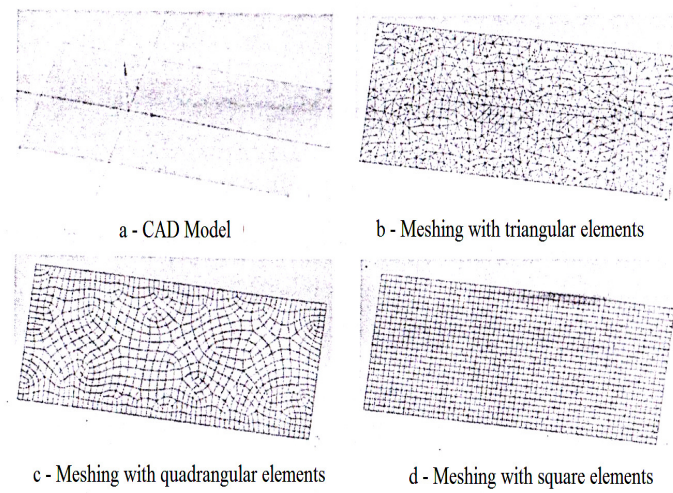

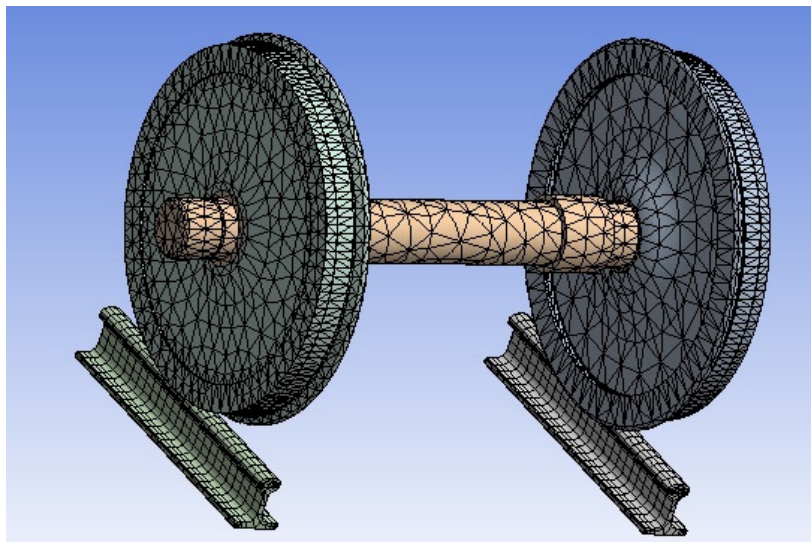**Fig.7 Wheel axle of CFR 060-DA Diesel-electrical locomotive.**

Fig.8 Wheel center of CFR 060-DA Diesel-electrical locomotive.

Fig.9 Wheel bandage of CFR 060-DA Diesel-electrical locomotive.

[3]

Mesh:

Structural Analysis programs perform simulations for models behavior by Finite Elements Method (FEM). The basic idea of FEM is that it is easier to find an approximate solution for structures behavior problems in large but finite number of points than it would be for exact analytical solution performed continuously on the whole structure in infinite number of points. Thus, programs which use FEM calculate structures behavior in net nodes of the mesh separately then according to the results obtained there, they express the whole structure behavior.

Fig.10 Combined model: a solid body with a flat plate; mesh variations.

a - CAD Model

b - Meshing with triangular elements

c - Meshing with quadrangular elements

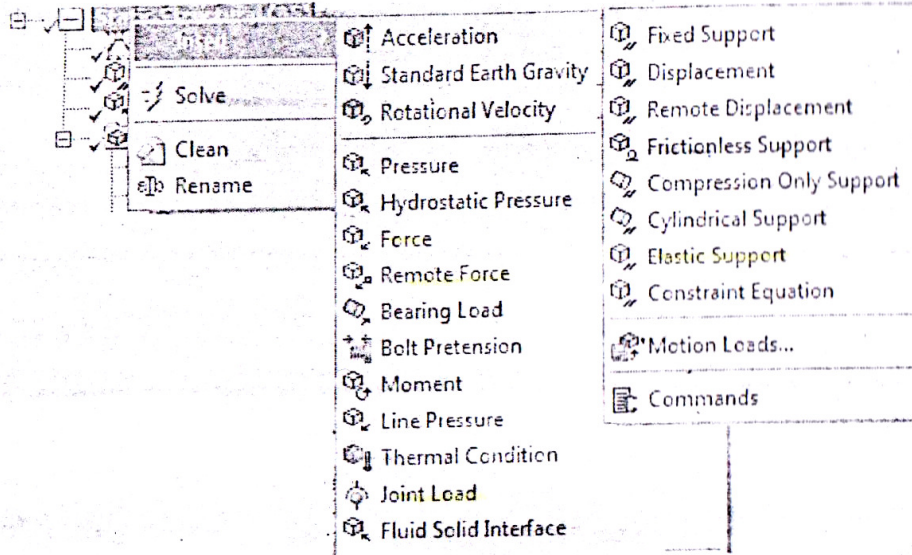d - Meshing with square elements

Fig.11 Variations of the mesh for a flat plate.
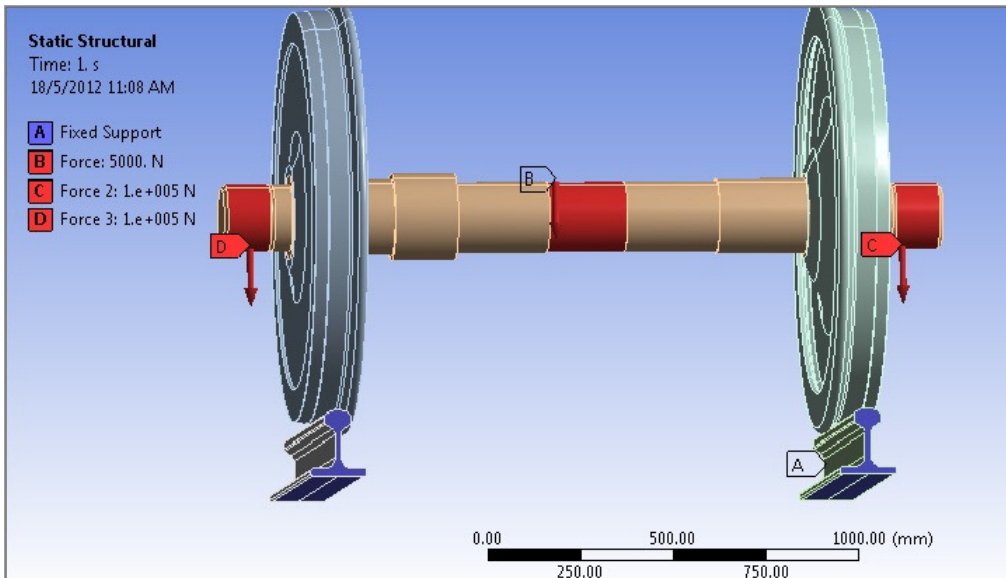

Fig.12 Wheelset and rails meshing.

Loads and Constrains:

**Fig.13 Active loads and constrains for a 3D model.**

In this stage, one can determine the followings: - Supports, - fixed support, - imposed displacements and rotations, - elastic supports.

In case of assemblies, in contact zone between two components, all types of kinetic joints can be defined.

Fig.14 Loads and constrains applied on the geometric model.

Results:

Specific deformations   Stresses   Predifined sets
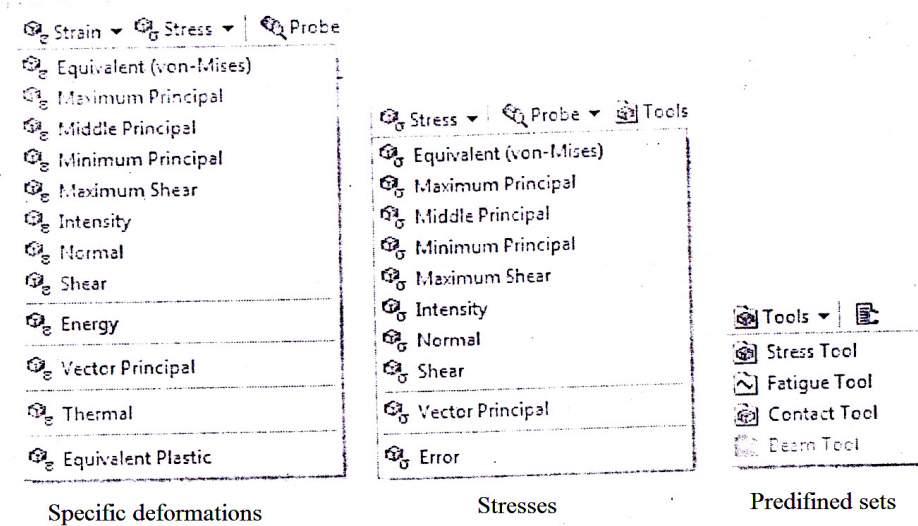
Fig.15 The results of static analysis.

The information related to contact can be determined from the predefined sets of results (contact tools):Contact status, stress caused by friction, the pressure in the contact zone, slip distance. In the text window (worksheet) fig.15, the information related to model solving appears. Images and animations saving is done with the tools which exist in the menu bar or the timeline window. This can be included in html report or can be saved as files with specific formats (animations avi, figures jpeg, tif, bmp, eps..).

Fig.16 Text window (solution information – worksheet).

Fig.17 Equivalent elastic strain, sectional view (the deformations are exaggerated).
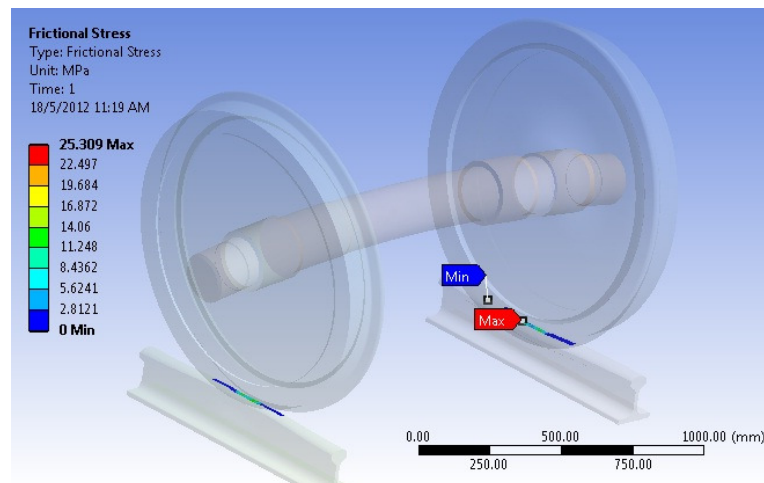
Fig.18 Frictional stress.
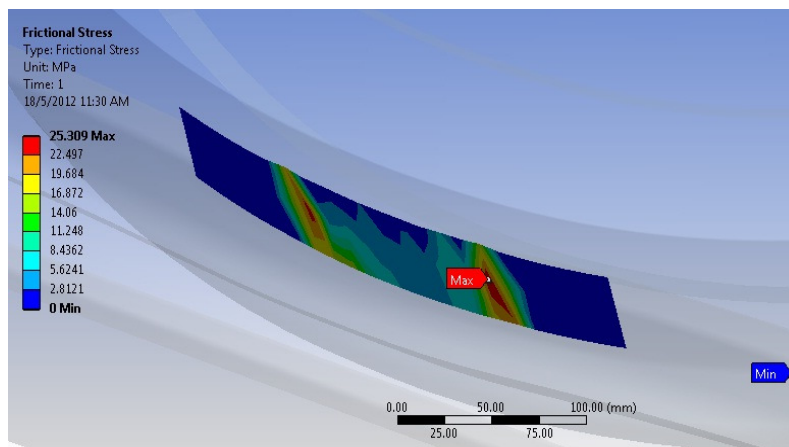
Fig.19 Frictional stress in the contact zone.

85

Fig.20 Total deformation (exaggerated).

Fig.21 Total exaggerated deformation ,sectional view.

Fig.22 Contact zone situation (status).
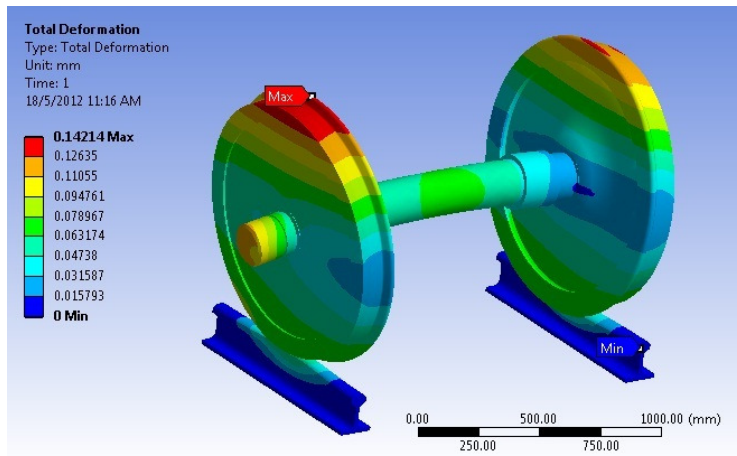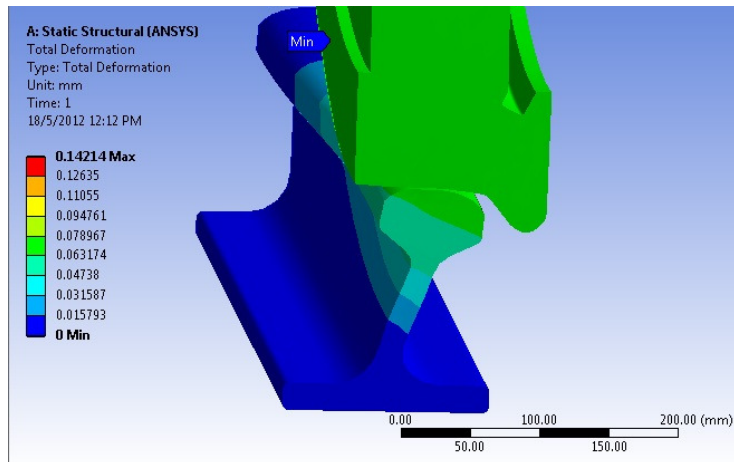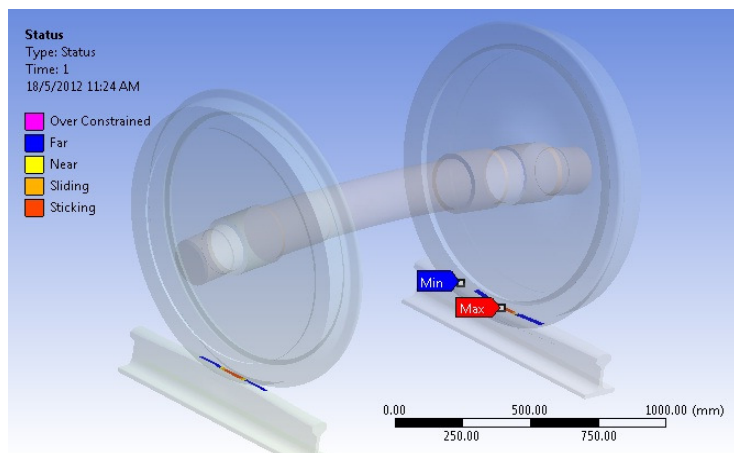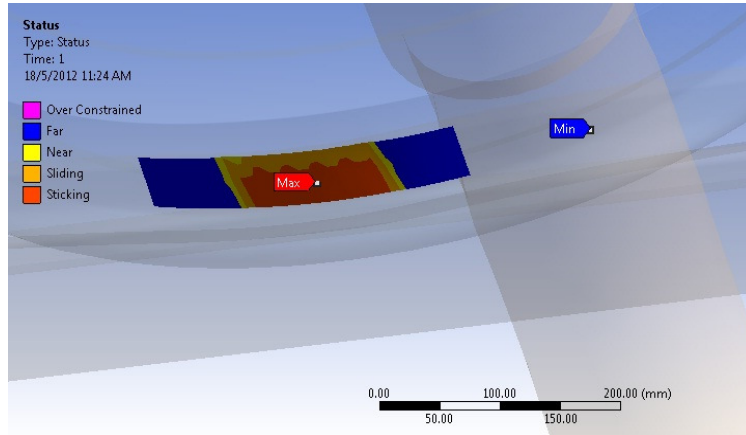
Fig.23 Contact zone situation – a close view.

Assemblies and contact regions :

Assemblies modulation in CAE environment contains creating finite elements for the interactions and contacts between components, defining work conditions of the participating parts whose behavior is the simulation target, those parts material characteristics determination, assemblies physical characteristics determination and solving method controlling. Mechanical module of workbench automatically detects the contact between the components of an assembly.

User can then edit the contact regions, or can define the parameters according to assembly type requirements. In connections menu there are two useful instruments for preliminary verification of assemblies modulation problems: contact tool and solution information.

Color code is:

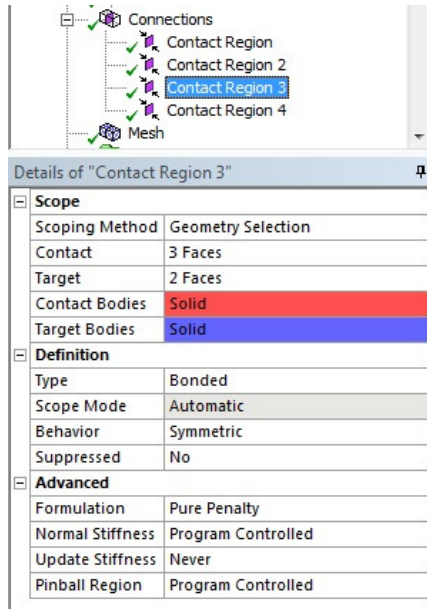|	
Red	The contact between the surfaces is open, but it's defined as closed.
Yellow	The contact is open.
Orange	The contact is closed, but has high amounts of holes/penetrations among elements.
Gray	The contact is inactive: this may appear with MPC contact formulation, Normal Lagrange contact or with auto asymmetric one.

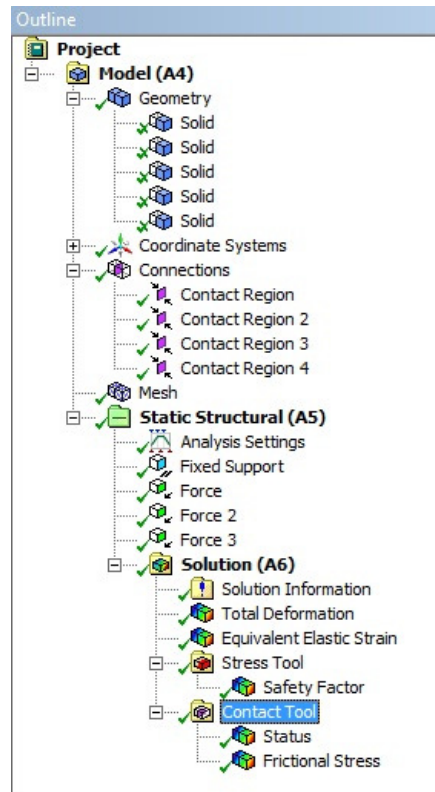**Fig. 24 Information and options of the without-separation contact.**
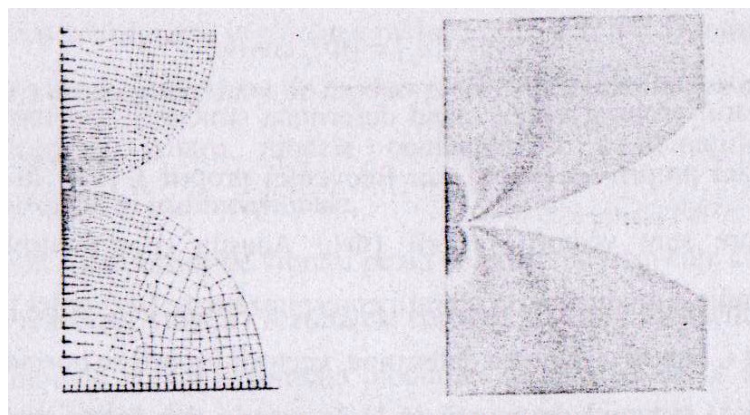
Fig. 25 Simulation contents.

Fig.26 Hertzian contact simulation.

Computer properties:

This simulation is performed on a Dell, Intel core 2 Duo computer with the following specifications:
Processor speed: 2.1,2.1 GHz double Dou cores.
Total RAM memory size: 4 Gbytes.
Video card capacity: 329 Mbytes.

4. RESULTS VERIFICATION

Generally, results verification is performed as follows:
a- Qualitatively: After processing, the model should be checked if it is acting as expected according to the requirements.
b- Quantitatively: Here, results precision is checked and to do that:
   1- The obtained results is compared with:
      o Experimentally obtained values.
      o Analytically obtained values.
      o Specialty literature results for similar situations.
   2- Mesh net is refined then the analysis process performed again, it can be said that the mesh is sufficiently fine when the mesh net is refined and the same results are obtained.
   3- The model can be studied within another program, but with the same mesh.
   4- In the case of assemblies, the information obtained in the contact zone after solving are useful, such as: contact status, penetration, due-to-friction tension or stress, etc.
   5- It is a must to check if the reaction forces are equal to the external forces.

5. CONCLUSIONS

The up-mentioned verification method shows that the applied simulation process on the wheel set chosen in this paper is qualitatively correct, while the quantitative verification is more difficult and needs lots of experiments to be performed on the same type of wheel set . for this purpose, by comparing the results obtained in this paper with the results obtained by Talamba, R. and Stoica,M. [4], it can be sufficiently accurate concluded that the simulation process is correct and the obtained results are close to reality. For comparison with analytically obtained results, a research paper done by Y.Zakaria entitled: "Creating a program in Matlab for the contact ellipse dimensions and friction coefficients analytically calculation" should be checked.
It can be noticed that the maximum deformation is located on a wheel, while the maximum stress is located on the other wheel and although the wheels are geometrically identical and identically loaded, however, there are differences between both of them related to the stresses, deformations and contact zone statuses. The differences are due to wheel set mass center deviation from the symmetry axis of the rails with the value y. This means that any deviation y leads to changes on the loads, stresses and deformations on both wheels, therefore, it leads to friction coefficient modifications, as a result, it modify also the traction coefficient. That is why the hunting

movement always accompanies the traction process. To check how the y distance is affecting contact between wheels and rail, y is modified and the results are registered for each value of y. Further simulations can be easily executed on the same created model. An external torque, for example, can be applied on the wheel set to study the contact status in traction mode.

Acknowledgment:

This work[1] is accomplished for the purpose of a PHD thesis fulfillment, which has the following title: "Contributions in the studying of modern Diesel-electrical locomotives traction performance" by Y.Zakaria.

Bibliography:

1- **Cristina Pupază; Radu Constantin Parpală :** *Modelare şi analiză structurală cu ANSYS workbench (Structural Modeling and Analysis using Ansys workbench)*, Editura Politehnica press, Bucureşti, 2011
2- **Ioan Sebeşan:** *Dinamica Vehiculelor de cale ferată (Dynamics of railroads Vehicle)*, Editura tehnică, Bucureşti, 1996.
3- **Ioan Zăgănescu:** *Locomotive şi Automotoare cu motoare cu ardere internă (Locomotives and self-propelled Vehicles with Internal Combustion Engines)*, Editura Didactică şi Pedagogică, Bucureşti , 1968.
4- **Talambă R.; Stoica M. :** *Osia montată (wheel set)*, Editura ASAB, Bucureşti, 2005.
5- http://www.keytometals.com
6- http://www.ansys.com

---

[1] Also available in Romanian language.