16th International Research/Expert Conference
"Trends in the Development of Machinery and Associated Technology"
TMT 2012, Dubai, UAE, 10-12 September 2012

STRESS ANALYSIS OF SPECIAL ELEMENTS FOR LINKS WITH CRANE RAILS

Savićević Sreten
Faculty of Mechanical Engineering
Džordža Vašingtona bb, Podgorica
Montenegro

Laković Dušan
Aluminium Plant Podgorica
Podgorica
Montenegro

Vukčević Milan
Faculty of Mechanical Engineering
Džordža Vašingtona bb, Podgorica
Montenegro

Janjić Mileta
Faculty of Mechanical Engineering
Džordža Vašingtona bb, Podgorica
Montenegro

ABSTRACT
The paper deals with rail clip, and comparation with experimental analysis. The main function of this element is providing relationship against turning rails. This paper is representing problem of finding stress state in the element with contact analysis. The purpose of this analysis is to identify differences in the results on actual examples which are based on the results of the finite element method (FEM) and experimental analysis. The results of numerical analysis are given graphically and numerically. The finite element meshes, model properties, stresses and position of gages are shown.
Keywords: Contact analysis, experimental analysis, FEM, rail clip

1. INTRODUCTION
The computer support in the design of elements and construction is often the shortest way to obtain images of the values and distribution of stress. Using the finite element method (FEM), many contact problems can be solved with relatively high accuracy [1].
The paper analyzes a model whose numerical results are obtained by finite element method. The main question that arises is - Why is the contact analysis applied to this model? The answer to this question will be obtained from the analysis of the topic of this paper. For evaluation of its validity the results of experimental analyzes can be used that serve only as a comparison, which are not included in the topic of this paper.

2. CONTACT ANALYSIS OF RAIL CLIP
In order to model the contact problem it is necessary to identify the parts that will be analyzed for their potential interaction. This step is extremely important because it is not known exactly where the contact will occur. For the good convergence of the solutions, the contact zones should be large enough in order to "capture" the contact [2].
In problems involving contact between the two limits, one of which is defined as a "target" and the other as a "contact" surface. All contact problems can be divided into two general classes: rigid - flexible and flexible - flexible. In addition to these classes, these types of contacts can also exist: point-to-point, point-to-surface and surface-to-surface. For rigid - flexible contact, target surface is always a solid surface and contact surface is a deformable surface. If all of the contact elements are on one surface and all of the target elements are on the other surface, then we can say that it is an

asymmetric contact. This is usually the most efficient way to model the surface-to-surface contact analysis and it is provided in this analysis [3].

In this analysis, the contact model is the type surface-to-surface and the class rigid - flexible. The main purpose of the application of such contact model is to obtain the equal displacements in the contact zone. Contact elements are arranged to oppose the penetration of the target surfaces. Regardless of this, the target elements can penetrate through the contact surface. Due to changes in the stiffness of the cross section in the loading zone, the displacements obtained by linear static analysis will not be in concordance with the real image of displacements [4].

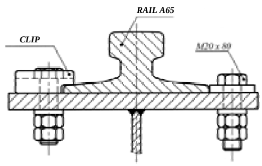

Figure1. Fit of the rail and the clip

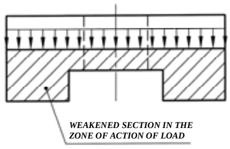

Figure 2. The crosssection in the loading zone

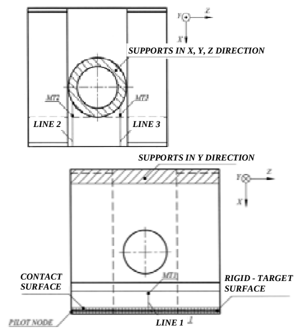

Figure 3. The support and contact pair with gauges

The load to which the clip is exposed is 11 KN. This load is transmitted to the clip through the line contact as shown in Figure 1. However, in the analysis, this load is shifted by 1 mm. The purpose of this is the emulation of the load provided in the experimental analysis on the area corresponding to the position and the size of the rigid - target surface in the contact analysis. This surface may be associated with the pilot node, which is really the element whose movement directs the moving of the target surface. The pilot node can be thought of as a ″ handle ″ of the target surface. Forces / torques or rotations / shifts can be assigned to the target surface through this node (element). The position of the pilot node is defined: the centre of mass of target surfaces, the existing point, the existing node, location in the coordinate system or the creation of additional nodes. The position of the pilot node is important only when the torques or rotations are applied. In this analysis, the load is applied through the pilot node that is connected to an existing point on the target surface (Figure 3.)

$\sum M_A = 0$, Assumption – the rail is absolutely stiff

$$F_{jah} \cdot u - F_{ah} \cdot v = 0$$

$F_{jah} = \frac{v}{u} \cdot F_{ah}$, F_{ah} =30.4[KN]

$$F_{jah} = 0.3621 \cdot F_{ah}$$

$F_{jah} = 11[KN]$ - the load of the rail clip

The rail clip is modeled by using 3D solid tetrahedral finite elements with 10 nodes (SOLID92 - Ansys) [5] and three degrees of freedom per node. Finite element mesh is made with higher density in the contact zone.

Ansys Software Package offers several types of contact methods (Penalty Method, Augmented Lagrangian Method, Pure Lagrange Multiplier Method, Internal Multipoint Constraint - MPC), by using these it is possible to model different contact situations such as penetration, the possibility / impossibility of separating the high / low contact strain. For the analysis of this model, the Augmented Lagrangian Method was used which presents an iterative series of Penalty Method. Contact traction (pressure and frictional stresses) are increased during the equilibrium iterations so that the final penetration is lower than the permissible one. Augmented Lagrangian Method is less sensitive to the size of the contact stiffness [6].

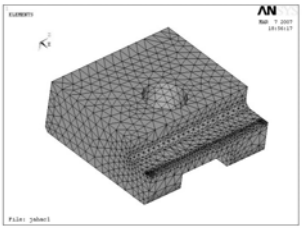

Figure 4. Element mesh

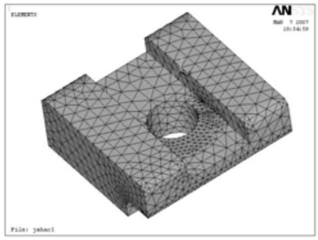

Figure 5. Element mesh in the hole

The contact task has 17 598 3D solid92 tetrahedral elements, 50 and 306 targe170 conta174 contact elements. The model is described with 26 971 nodes and 79 478 degrees of freedom.

3. RESULTS OF THE CONTACT ANALYSIS

The calculated values of stress and strain provide the basis for the assessment of the accuracy of the results of contact analysis. For this assessment, the results of experimental analysis will also serve.
The directions of strain measurements in the contact analysis correspond to the directions of the strain measurements for all gauges respectively. The values of strain are determined by interpolation through nodes which belong to the lines 1, 2 and 3 and to which the gauges 1, 2 and 3 also belong (Figure 3.).

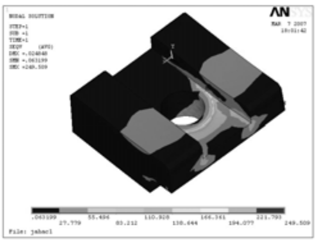

Figure 6. Von Mises streses in the hole zone

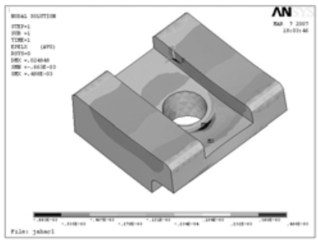

Figure 7. Strains in x direction in the hole zone

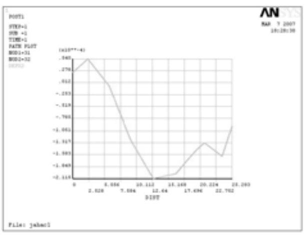

Figure 8. Strains in x direction for gauge 1

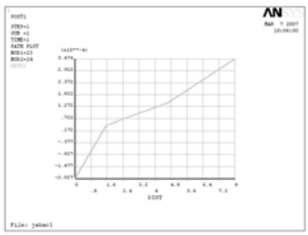

Figure 9. Strains in x direction for gauge 2

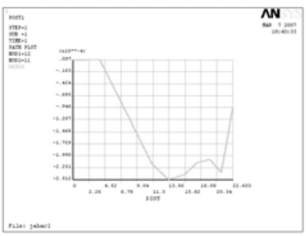

Figure 10. Strains in x direction for gauge 3

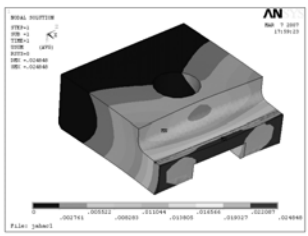

Figure 11. Total shifts

For the evaluation of the results of analysis in engineering practice, the two criteria can be used, namely: Mises' and Treskin's. However, neither of these criteria is completely satisfactory, although Mises' criterion agrees better with the experimental results. For this purpose, for the evaluation of distribution and the stress value the Mises' criterion is taken, regarding the foreseeable elastic behavior where the most of the energy is provided by loading, used for geometrical deforming.
Since the element includes the three-dimensional (spatial) stress state, the value of deformation in x direction (direction of strain measurement in the experiment) are determined. Numerical analysis is performed for the loads given in Table 1.
The clip material is Č0561 (σ_T = 345 N/mm^2, E = 206 000 N/mm^2, ν = 0.3).

Table 1. Comparison of the experimental and numerical results

	Stresses [N/mm^2]					
Loads [N]	Gauge 1	Gauge 2	Gauge 3	Gauge 1	Gauge 2	Gauge 3
2518.508	16.7764	-10.2439	-14.6126	15.1615	-9.7241	-11.4964
5025.167	29.3117	-19.0791	-26.5873	30.2517	-19.4023	-22.9386
7000.667	37.7345	-25.6081	-35.3722	42.1443	-27.0297	-31.9563
9013.144	46.8141	-32.9296	-45.0445	54.2595	-34.7999	-41.1427
10953.51	59.5773	-39.7719	-55.2629	65.9406	-42.2918	-50.2455
Experimental results				Numerical analysis with Ansys		

4. CONCLUSION

From the table, the apparent discrepancies of results can be noticed and they are mostly prominent in the gauge number 1. Since the gauge is in the zone of stress concentration and small variations of the position (measured value of the position of gauge) may affect the accuracy of numerical results. This also applies to other gauge in which this phenomenon is less prominent. The results are certainly affected by the other phenomena that are not modeled within the contact analysis and occur during the experimental analysis, such as friction and sliding on the clip supports, strain of the tool elements (reducing the value of the energy incorporated in the clip load) for testing and similar. It is clear that when comparing the numerical and experimental results, we cannot talk about their concordance rather then values discrepancies. Loads values in Table 1. correspond to the load steps from the experiment. This analysis is applied with the aim of determining the strain at points corresponding to the position of the gauges.
The evaluation of stress value is determined by Von Mises'stress. Although these stresses are in the zone permissible for a given material, there is the possibility of optimizing the geometry (mass). Reducing the depth of groove the stiffness of the clip increases and thus the possibility of reducing the other dimensions. This paper shows that the finite element method has indispensable role when performing the contact simulation.

5. REFERENCES

[1] Savićević S., Janjić M.: Experimental and FEA Research of Stresses on Elements of Helicoidal Shell Shape. 11th International Research/Expert Conference - TMT 2007, Hammamet, Tunisia, 2007. ISBN 978-9958-617-34-8, pp 963-966.
[2] Lubarda V.: Otpornost Materijala, Titograd, 1989. godine
[3] Yan Gu, Wen Chen, Chuan-Zeng Zhang: Singular Boundary Method for Solving Plane strain Elastoplastic Problems, International Journal of Solids and Structures 48 (2011) 2549-2556.
[4] Yan Gu, Wen Chen, Jinyang Zhang: Investigation on Near-Boundary Solutions by Boundary Method, Engineering Analysis with Boundary Elements 36 (2012) 1173-1182.
[5] ANSYS – Release 8.0, Theory reference
[6] Vukčević M., Janjić M., Šibalić N.: Stress FEM Simulations of Axis Symmetrical Element. 12th International Research/Expert Conference - TMT 2008, Istanbul, Turkey, 2008. ISBN 978-9958-617-41-6, pp 1081-1084.