Finite element analysis of fatigue behavior of Duplex treated EN 34CrNiMo6 steel

R.M. Buzdugana [b*], **I. Mitelea** [a], **C. Cioana** [a]

[a] Politehnica University of Timişoara, Materials and Manufacturing Engineering Department, Timişoara, Romania
[b] National R&D Institute for Welding and Material Testing - ISIM Timisoara, Romania

E-mail: mdobra@gmail.com

Keywords

Finite Element Analysis, Duplex treatments, nitriding, fatigue behavior

1. Introduction

Finite Element Analysis (FEA) is based on that an approximate solution to any complex engineering problem can be reached by subdividing the problem into finite elements. Using this method, complex partial differential equations that describe the behavior of structures can be reduced to a set of linear equations that can easily be solved using matrix algebra. The method is used in many engineering areas, such as aerospace, automotive, biomedical, chemical, electronics and plastics industries. In addition, it has been used extensively for forensic investigations [1, 2].

Like any other numerical approximation method, the solutions that are produced by finite element analysis contain some errors also. The magnitude of the errors is dependent on the type, size, and fineness of the used model. That is the reason why not all finite element models are created equal. Indeed, the industrial experience is the most critical factor in obtaining accurate results [3]. Processes that use FEA involves carrying out a sequence of steps in some way.

Those sequences take two configurations, depending on (i) the environment in which FEA is used and (ii) the main objective: model-based simulation of physical systems, or numerical approximation to mathematical problems [4]. A common use of FEA is represented by the simulation of physical systems, which requires models of such systems. Consequently, the methodology is often called model-based simulation. The process is illustrated in Figure 1 [5]. The centerpiece is the physical system to be modeled. The process of discretization is carried out to produce the discrete model. The solution step is handled by an equation solver often customized to FEA, which delivers a discrete solution (or solutions).

The paper presents the finite element analysis of Duplex treated EN 34CrNiMo6 subjected to rotating bending fatigue.

2. Experimental procedure

The finite element analysis was performed on untreated EN 34CrNiMo6 and Duplex treated by gas nitriding and work hardening steels specimens. The finite element analysis using the CosmosWorks FEA software was used to determine the stress concentration or the critical dimension, and to determine the nominal stress applied for life prediction. The analysis started from the 3D geometrical model of the specimen used in rotating bending fatigue test (Figure 2).

A much finer mesh was selected in areas where the stress evolution is faster and of higher interest (in the calibrated area). In Figure 3 is shown the mesh structure of the 3D model of the specimen used in rotating bending fatigue test. The mesh consists in 96334 elements and 141780 nodes.

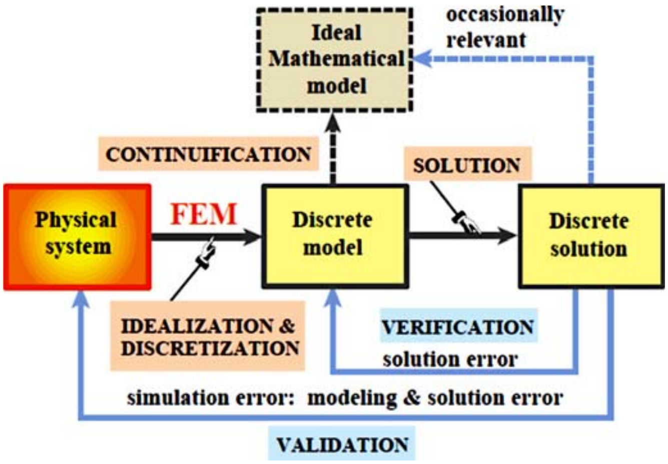

Figure 1. The Physical FEA [5].

Figure 2. 3D model of the specimen used in rotating bending fatigue test.

Figure 3. Meshing model used in rotating bending fatigue test.

Loads and restraints are necessary to define the service environment of the model. The results of analysis depend directly on the specified loads and restraints. Loads and restraints are applied to geometric entities as features that are fully associative to geometry and automatically adjust to geometric changes. The model was constrained at one end and loaded with a constant force at the other end, see Figure 4.

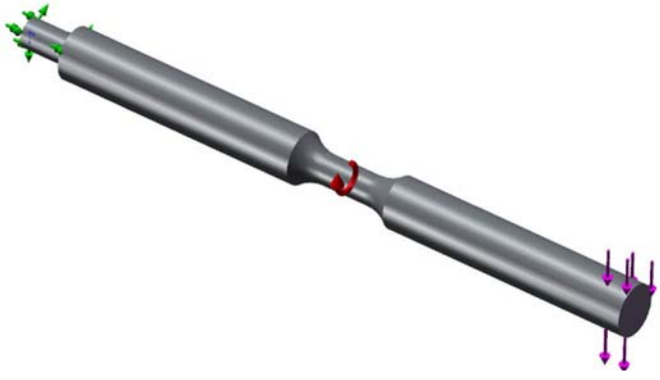

Figure 4. loads and restraints.

3. Results and discussions

The stress distribution in the central area of untreated specimens is roughly symmetrical, with a region characterized by maximum stress of σ_{max} = 689 MPa and regions with minimum stresses of σ_{min} = 327 MPa (Figure 5).

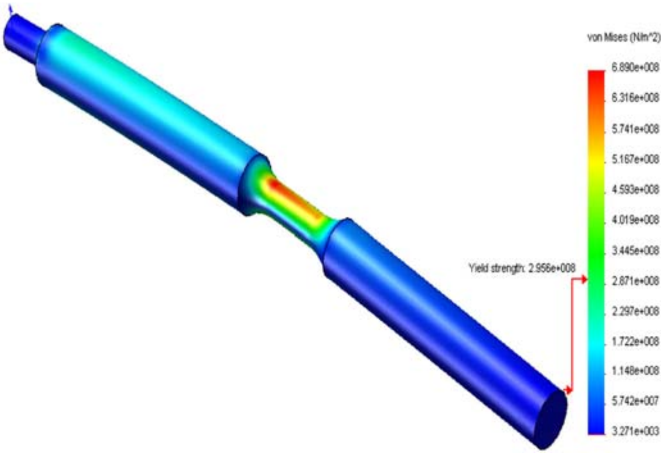

Figure 5. Von Mises stress distribution plot for untreated specimens.

The resultant displacements plot and the equivalent strain plot for untreated specimens are presented in Figure 6 and Figure 7.

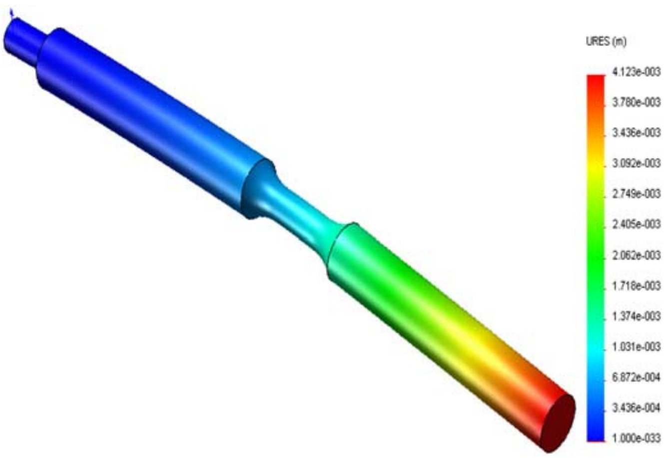

Figure 6. displacements Plot.

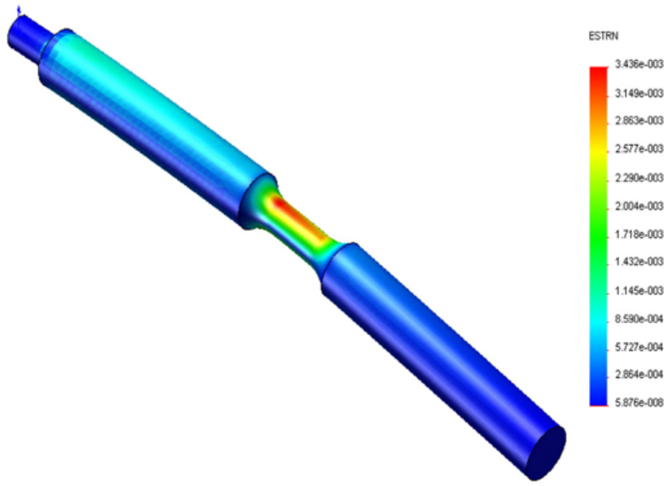

Figure 7. Equivalent Strain Plot.

The total life plot is displayed in Figure 8 and shows that failure due to fatigue is likely to occur in the calibrated region after approximately 24895 cycles.

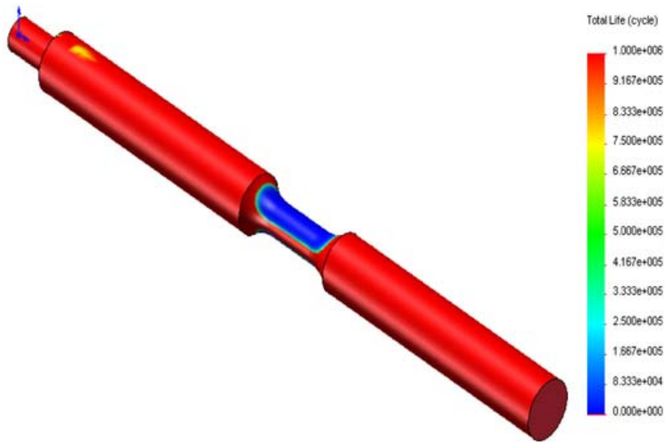

Figure 8. Life Plot.

The stress distribution in the central area of the Duplex treated specimens is presented in Figure 9.

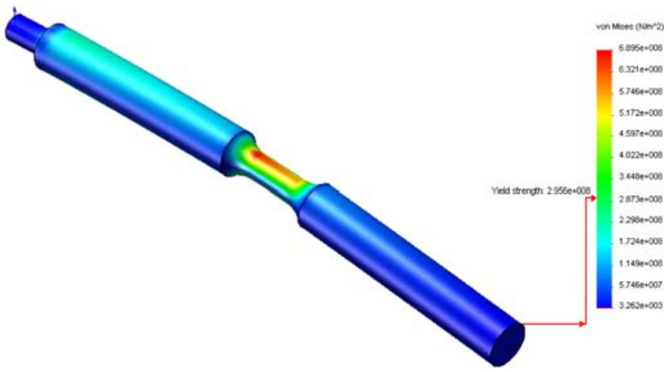

Figure 9. Von Mises stress distribution plot for Duplex treated specimens.

The resultant displacements plot and the equivalent strain plot for Duplex treated specimens are presented in Figure 10 and Figure 11.

The total life plot is displayed in Figure 12 and shows that failure due to fatigue is likely to occur in the calibrated region after approximately 52462 cycles.

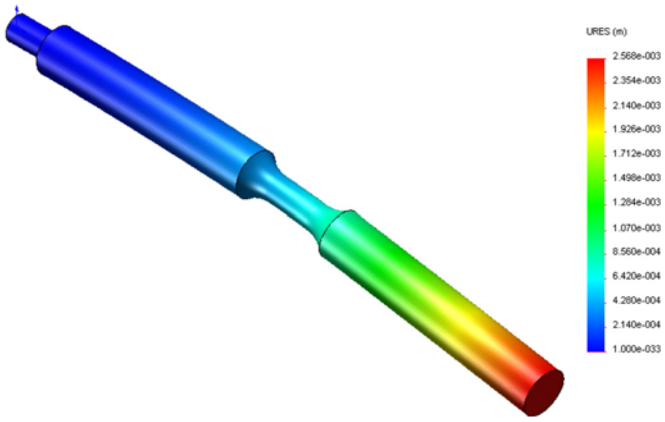

Figure 10. displacements Plot for Duplex treated state.

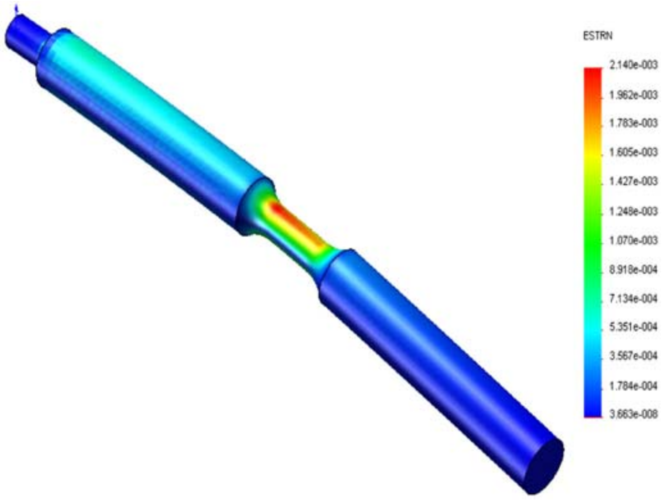

Figure 11. Equivalent Strain Plot for Duplex treated state.

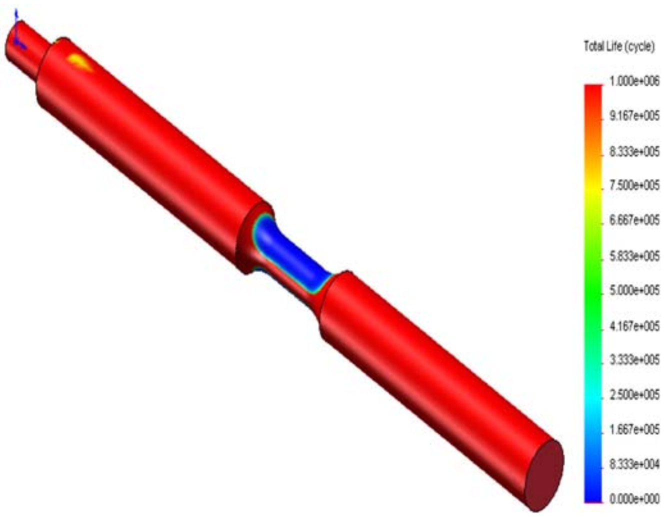

Figure 12. Life Plot.

The results obtained by FEA are presented in Table 1.

Table 1. Von Mises Stress and Life cycles values.

Structural state	Von Mises Stress	Life cycles
Untreated	400	24895
Duplex treated	690	52462

4. Conclusions

The finite element analysis was performed on EN 34CrNiMo6 untreated and Duplex treated steels specimens using CosmosWorks FEA software. The method was used in order to evaluate the life time of untreated and treated materials.

After performing the simulation, it can be concluded that the results obtained through Finite Element Analysis for the Duplex treated state are better compared to untreated state.

Acknowledgement

This work was partially supported by the strategic grant POSDRU/88/1.5/S/50783, Project ID50783 (2009), co-financed by the European Social Fund – Investing in People, within the Sectoral Operational Programme Human Resources Development 2007 – 2013.

References

[1]. Radu, B., Codrean, C., Finite element modeling of thermal field developed in induction brazing of stainless steels with quasi-amorphous brazing alloys, M.T.P., IXth Edition Timişoara academic Days International Symposium, May 26-27, 2005;

[2]. http://www.colorado.edu/engineering/cas/courses.d/IFEM.d/;

[3]. http://www.sv.vt.edu/classes/MSE2094_NoteBook/97ClassProj/num/widas/history.html

[4]. Wang, L. et.al., Fatigue life analysis of aluminum wheels by simulation of rotary fatigue test, Journal of Mechanical Engineering, Vol. 57, 2009;

[5]. Dumitru, I., Faur, N., Elemente de calcul şi aplicaţii în rezistenţa materialelor, Timisoara, 1997