Applied Computer Science, vol. 11, no. 4, pp. 20–29

Submitted: 2015-09-16
Revised: 2015-10-30
Accepted: 2015-11-26

Abaqus, xFEM, numerical analysis, crack propagation

Patryk RÓŻYŁO[*]

NUMERICAL ANALYSIS OF CRACK PROPAGATION IN A STEEL SPECIMEN UNDER BENDING

Abstract

The paper compares numerically modeled crack propagation in a steel specimen with a real process of fiber separation. The objective of the study was to perform numerical analysis of crack propagation in order to determine the shape of a crack and the distribution of stresses in the entire model. A CAD model of the test specimen was prepared based on geometric parameters determined for a real model. The numerical analysis was performed using the computer simulation program Abaqus 6.14. The experiment of specimen bending was conducted using bench tools. Crack propagation was simulated by the numerical method xFEM which enabled visualization of fiber separation in the test specimen. The numerical results of crack propagation were compared with the experimental findings about cracking in a real specimen.

1. INTRODUCTION

The process of permanent fiber separation is nowadays a common problem in operation of machines. The essence of improving operational conditions of machines lies in reducing manufacturing and structural defects of their subassemblies in order to avoid undesired problems in the future. Given their long-term operation, it is impossible to avoid occurrence of undesired processes that will have a negative effect on operation of these machines. The real degradation of their technical condition is often caused by temporary exceeding

---

[*] Lublin University of Technology, Nadbystrzycka 36, 20-618 Lublin, +48 603 359 217, p.rozylo@pollub.pl

of both maximum fatigue strength and experimentally determined maximum strength of material. The most popular steel subassemblies are exposed to overload and failure on a nearly constant basis.

Crack propagation in machine components is a very complex problem. During their operation, structural members can be subjected to loads, which can lead to partial or total material decohesion. The current problems of crack mechanics were described in the study [4], which discusses the problem of microcrack development leading to element failure.

Cracking is defined as partial or total separation of material due to applied loads. The process of cracking is typically divided into three stages. The first stage of crack occurrence is the initiation of defects in material structure of a given member. Another stage consists in gradual development and combination of the initiated defects due to cracking, whereas the final stage involves the occurrence of the main crack that leads to failure of the structural member [7].

Crack propagation processes in steel machine components can be investigated by numerical analysis such as the finite element method (FEM).

The correctness of simulating real processes by numerical methods depends on defining their boundary conditions. It is also important to possess both the knowledge about essential properties of materials and the skill of defining interactions between implemented models. Also, what plays a significant role in crack analysis is the preparation of a finite element mesh (by xFEM).

Specialist simulation software enables predicting strains and stresses inside machine components by simulating real processes which occur when these components are being operated. The authors of the studies [5, 6] describe the application of finite element method to selected problems of engineering constructions mechanics.

The publications such as [2, 3, 8, 9, 10, 11] deal with advanced problems of crack propagation in materials resulting from the applied boundary conditions and loads acting on these materials.

Advanced systems for numerical computations such as ABAQUS allow us to generate a finite element mesh, thereby enabling us to simulate failure caused by permanent separation of material fibers.

The finite element method has a positive effect not only on improving properties of structural subassemblies, but also on reduction of stresses. In the long run, the reduction of material effort leads to longer and defect-free operations of the subassemblies. Correct analysis of structure design requires specific knowledge, skills, experience and time. Only in this way can defects of design be avoided in the future.

2. MATERIALS AND METHODS

The tests were performed on a C45 steel rectangular specimen with symmetric cut-outs in the centre. The numerical model of the real object was designed in compliance with the real specimen's geometric dimensions using Abaqus 6.14. The specimen was described by mechanical properties which are listed in a table given below.

Tab. 1. Characteristics of C45 steel [1]

Material: Steel C45	
Young's Modulus [MPa]	210000
Poisson's Ratio	0.3
Yield Strength [MPa]	360
Tensile Strength [MPa]	610
Elongation [%]	15.5

The numerically modeled element was assigned plastic-elastic properties. The tested model was characterized by relatively small overall dimensions. The main aim of the research was to determine the shape of fiber separation for the investigated cracking problem. The sample was designed with basic mechanical properties due to static analysis.

Since the numerical computations only involved the problems of statics, neither time nor specimen weight were examined.

The geometric dimensions of the object are given in mm, while the specimen itself has a nominal thickness of 1 mm. The real and numerical models are shown in the figures given below.

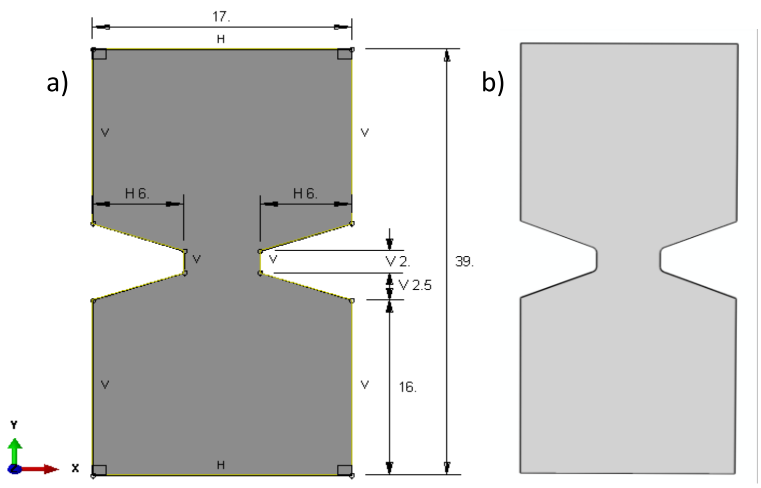

Fig. 1. Test specimen: a) geometric dimensions, b) numerical model with internal fillets at 0.5 mm narrowing [source: own study]

Fig. 2. Real specimen [source: own study]

The numerical model entirely reflected the measurements made on a real element. The numerical model was described by suitably defined boundary conditions which complied with the real process of crack propagation. Some part of the external upper and lower surfaces were totally fixed (all degrees of freedom were blocked) in the spots indicated in a figure below. The lower part of the specimen’s flank was subjected to load in the form of 10 MPa pressure. The pressure was distributed uniformly over the entire surface and acted in the opposite direction to X of the global system of coordinates, which led to specimen bending. The boundary conditions describing the test specimen are shown in a figure below.

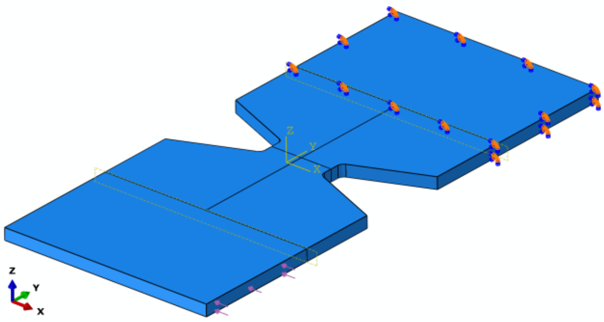

Fig. 3. Boundary conditions of the numerical model [source: own study]

The discretization of the numerical model was performed for the highest possible preparation of a finite element mesh. The numerical model of the specimen was partitioned in order to obtain the most accurate possible type of FEM mesh. The partitioning was continued until a hexagonal mesh was generated. This kind of mesh yields optimal results for solid models. The finite element mesh was made up of hexagonal elements using the Sweep mode (which is typically used for simple geometry elements). The elements were made more dense relative to the central axis of the entire object. The object was modeled using C3D8R elements with 10998 mesh elements and 15332 nodes. The C3D8R elements with three degrees of freedom and reduced integration produce highly accurate results because false forms of strain are eliminated due to the application of higher order polynomial equations [12]. The developed numerical model and the finite element mesh are shown in a figure below.

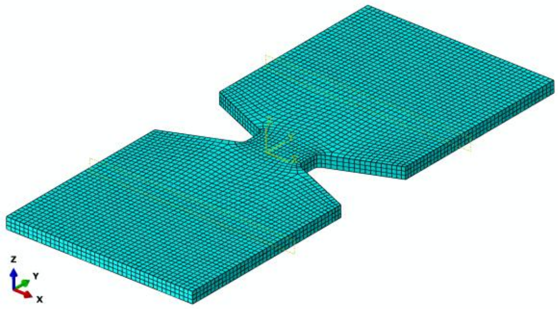

Fig. 4. Numerical model with a FEM mesh [source: own study]

The mesh density was increased not only globally but also locally in order to obtain the best results possible. To this end**,** three mesh elements were generated over the thickness of the entire model to obtain more precise analysis results. When using the finite element method, it is important that the meshing and mesh density be prepared in a correct way in order to prevent false results and stress concentration in unexpected regions. Crack propagation was initiated in Abaqus using the xFEM method (to initiate and propagate processes of material fiber separation) for the entire numerical model. The real model was mounted in a vice and subjected to bending with bench tools until material cracking.

3. RESULTS

The numerical results reflect the stages of cracking and enlargement of the crack's shape. The form of crack propagation was modeled numerically by the xFEM method. Based on critical stresses (when the material suffered permanent specimen failure), the program indicated the region and direction of crack initiation in the model. Implemented in numerical environment, the advanced xFEM method allows us to simulate crack initiation and its development once the parameters of the analyzed process have been precisely defined. The results of crack propagation in the test specimen as well as increases in stresses and local enlargement of the simulated crack are shown in figures below.

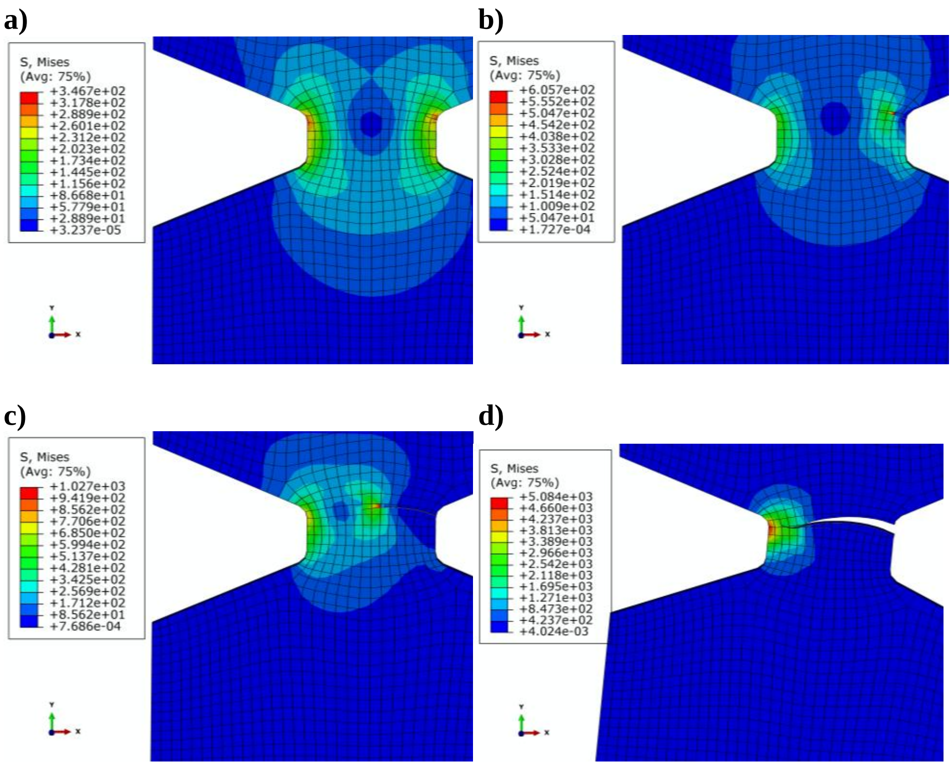

Fig. 5. Crack initiation and propagation [source: own study]

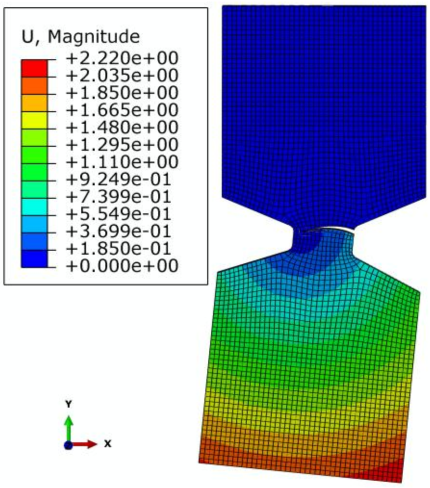

Fig. 6. Distribution of material displacement during cracking [source: own study]

Crack initiation occurs exactly in the narrow region of the specimen, right at the fillet radius. Using FEM analysis, it was possible to define a relationship between the crack initiation point and the increase in load.

The above characteristics and images of the tested point are shown in a figure given below.

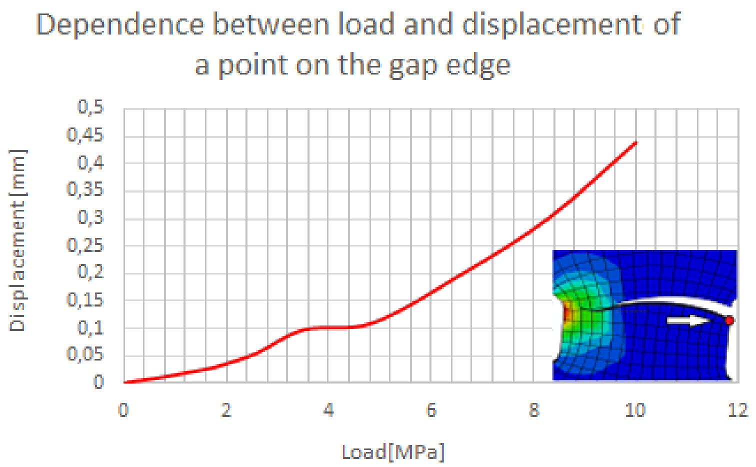

Fig. 7. Load characteristics versus crack initiation point displacement [source: own study]

The width of the narrowing is 5 mm. The total length of the produced crack is practically as wide as the narrowing.

Initially, when the applied load ranges from 0 MPa to 3.5 MPa, the displacement of the crack initiation point increases almost linearly compared to low displacements of about 0.1 mm due to specimen deformation. When the applied load ranges from 3.5MPa to 5 MPa, cracking occurs yet practically no displacement can be observed. From about 5 MPa onwards, the point displacement dramatically increases due to the proceeding separation of material fibers. At the highest load of 10 MPa, the displacement of the point within the initiated crack is almost 0.45 mm, which corresponds to the maximum displacement observed for the numerical model.

The shape of the crack resembles an arch described by a similar radius over the predominant length of the crack. The subsequent part of this paper will present images of the simulated distribution of displacements in the specimen due to the applied load.

a) b)

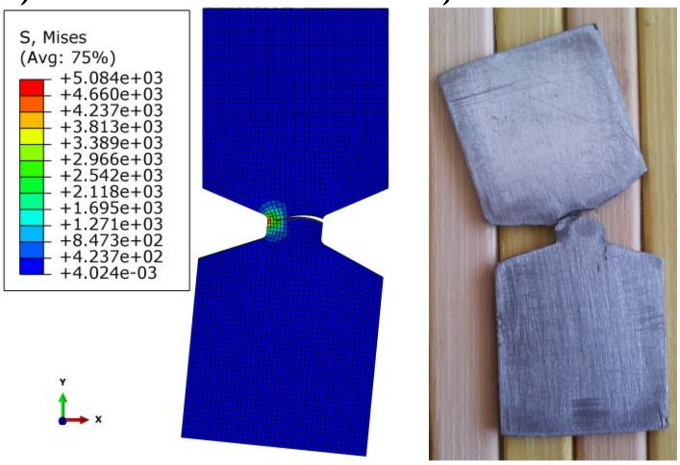

Fig. 8. Comparison of cracking in numerical (a) and real specimens (b) [source: own study]

The numerical results can be used to illustrate the shape of crack that resembles that in the real specimen. The shape of the enlarging opening initiated in the upper part of the specimen's narrow region just before the minimal fillet radius. Further crack propagation in the form of an enlarging opening occurred along the narrowing where stresses temporarily increased and their values significantly exceeded the assumed steel strength.

The cracking of the numerically prepared specimen initiates at the moment of excessive material effort at about 350 MPa, i.e. at the moment when the first nodes got separated in the narrow region of the specimen. The reaching of strength limit only initiated the cracking process; however, crack propagation occurred with gradual loading of the numerical model. The cracking continued until producing a full opening that could be simulated by the Abaqus program with constant support of the xFEM method.

At the final stage of crack propagation, the value of reduced stresses is exactly 5084 MPa (the program writes it as 5.084e+03, which should be read as $5.084*10^3$); therefore, it exceeds the element material's strength by several times.

The results enable performing further optimization analyses for similar specimens aimed at reducing both their weight and stresses. The simulations were performed based on predefined boundary conditions, which made it possible to obtain correct results in the form of distribution of stresses in the FEM model. The numerical analysis indicated regions of reduced stresses and displacements in the test specimen, and revealed crack enlargement. The test provided information about the degree of material effort in the test specimen and enabled visualization of the crack propagation process.

4. CONCLUSIONS

Finite element method enables us to analyze complex physical and mechanical advanced processes via complicated computations which solve real problems.

The numerical analysis revealed that – at long or excessive operation – loads exceeding maximum allowable strength exert a negative effect on the specimen, usually leading to its undesired, irreversible failure.

The FEM results demonstrate that specimens must be correctly and carefully prepared and subjected to strength testing in order to reduce their susceptibility to external loads.

Nowadays the problems of advanced engineering pertain to preventing material, structural and production defects. Numerical analyses enable us to predict the behavior of structural members and mechanisms of processes already at the stage of their design.

The modeling of complex processes which simulate real physical processes and mechanical behaviors helps prolong service life of produced mechanism and eliminate defects of design.

Summing up, the FEM results have led to the following conclusions about the investigated process:

- with correctly defined boundary conditions it is possible to obtain detailed information about mechanical and physical processes which occur during operation of machines,
- finite element method and advanced xFEM algorithms enable producing material cracking results that model real processes of crack occurrence already at the stage of design prior to the production of machine components,
- xFEM results of crack propagation help prevent technological and manufacturing problems in the future,
- the kind of load and the place where it is applied have a significant effect on the shape of the crack and crack initiation region,
- the characteristics of crack initiation and propagation regions enable us to determine the impact of external loads, progressing displacements and deformations.

REFERENCES

[1] BANASZEK J.: *Examples of calculations within machines constructions basics Part II*. The University Publishing House, 1996, p. 196–197.

[2] GAJEWSKI J., NOWAKOWSKI P., RÓŻYŁO P.: *Prognozowanie występowania pęknięć w oparciu o system FEM-MLP*, Logistyka 2015, nr 3, s. 1381–1386.

[3] GILL P., DAVEY K.: *Analysis of thermo-mechanical behaviour of a crack using XFEM for Leak-before-Break assessments,* International Journal of Solids and Structures, 2014, vol. 51, p. 2062–2072.

[4] JONAK J.: *Zagadnienia mechaniki pękania i skrawania materiałów.* Lublin: Politechnika Lubelska, 2010, s. 90–95.

[5] KĄKOL W., ŁODYGOWSKI T.: *Metoda elementów skończonych w wybranych zagadnieniach mechaniki konstrukcji inżynierskich*. Politechnika Poznańska, 2003.

[6] KLEIBER M.: *Wprowadzenie do metody elementów skończonych*, Biblioteka Mechaniki Stosowanej IPPT PAN, PWN, Warszawa-Poznań, 1985.

[7] MOES N., BELYTSCHKO T.: *Extended finite element method for cohesive crack growth*. USA: Northwestern University, 2001.

[8] NASIRMANESH A., MOHAMMADI S.: *XFEM buckling analysis of cracked composite plates*, Composite Structures, 2015, vol. 131, p. 333–343.

[9] Naderi M., Iyyer N.: *Fatigue life prediction of cracked attachment lugs using XFEM*, International Journal of Fatigue, 2015, vol. 77, p.186–193.

[10] RÓŻYŁO P., GAJEWSKI J., NOWAKOWSKI P., MACHROWSKA A.: *Zastosowanie sieci RBF w analizie pękania elementów maszyn*, Logistyka 2015, nr 3, s. 4172–4186.

[11] RÓŻYŁO P., WÓJCIK Ł.: *Comparison of numerical and experimental analysis of the crack propagation process*, Applied Computer Science 2015, vol. 11, no. 2, p. 60–67.

[12] ZIENKIEWICZ O.C., TAYLOR R. L.: *Finite Element Method (5th Edition)*. Vol. 2, Solid Mechanics, 2000, Elsevier.