

Optimization of Element Type for FE Model and Verification of Analyses with Physical Tests

M. Tufekci, C. Guven

Abstract—In Automotive Industry, sliding door systems that are also used as body closures are safety members. Extreme product tests are realized to prevent failures in design process, but these tests realized experimentally result in high costs. Finite element analysis is an effective tool used for design process. These analyses are used before production of prototype for validation of design according to customer requirement. In result of this, substantial amount of time and cost is saved.

Finite element model is created for geometries that are designed in 3D CAD programs. Different element types as bar, shell and solid, can be used for creating mesh model. Cheaper model can be created by selection of element type, but combination of element type that was used in model, number and geometry of element and degrees of freedom affects the analysis result. Sliding door system is a good example which used these methods for this study. Structural analysis was realized for sliding door mechanism by using FE models. As well, physical tests that have same boundary conditions with FE models were realized. Comparison study for these element types, were done regarding test and analyses results then optimum combination was achieved.

Keywords—Finite Element Analysis, Sliding Door Mechanism, Element Type, Structural Analysis.

I. INTRODUCTION

FINITE element method is a numerical approach which can be used to solution of engineering problems for many different engineering disciplines which as heat transfer, electromagnetism, fluid mechanics etc. S. Piskin has said that modern finite element method is started in the early 1900s. In the beginning, investigators were modelled by using elastic pieces which are the same with each other, for elastic situations [1]. Richard Courant used to segmented polynomial interpolation on the triangular sub-regions (elements) for solution of torsion problems in his article that was published in 1943 [2]. So we can say that he is first scientist who has developed finite element method. Finite element method was used by American and European aircraft industries in the 1950s and it was developed day by day. The name 'Finite element method' was first used by Clough [3]. In the beginning, finite element method equations was solved by humans so models had a few element (10-100). This factor affects sensibility of solution. But then equations was transformed that you can solve by computer and number of

M. Tufekci is with Rollmech Automotive San. ve Tic. A.S., Bursa, 16140 Turkey (phone: +90 549 800 04 89; fax: +90 224 243 87 67; e-mail: mustafa.tufekci@rollmech.com).

C. Guven is with Rollmech Automotive San. ve Tic. A.S., Bursa, 16140 Turkey (phone: +90 549 800 04 72; fax: +90 224 243 87 67; e-mail: caner.guven@rollmech.com).

element was increased. Nowadays, a civil plane's FE models may have 10 million elements [1].

Finite element method gives solution approximately to us. Model can be resolved after changing some parameters as element's type, number, and shape, contact's type thereby differences between solutions can be observed. This process's name is optimization. In this paper, effect of element's type for solution was investigated. Shell and solid elements can show different behavior in analyses to us [4]. Analyses were solved for 4 different element types as 2D shell tria and quad, 3D solid tetra and hex. Product that was chosen, is mechanism bracket of sliding door system produced at Rollmech Automotive, Bursa. Analysis results were correlated with physical test that has same boundary conditions. Finally the results were compared and the analysis that is the closest to real was seen. In this way, optimum element type was determined.

II. METHOD

First of all, product that we investigate on was chosen. Product data should be modelled by 2D or 3D elements. So product was determined as sheet metal. Product that we choose is mechanism bracket of sliding door upper system. Product data was shown in Fig. 1.

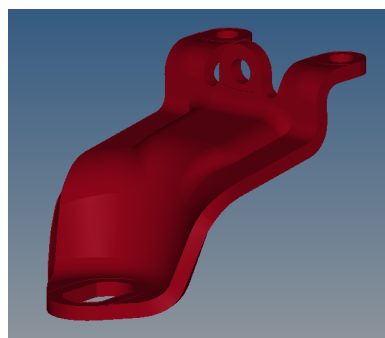


Fig. 1 Mechanism Bracket CAD Data

Product's thickness is 5 mm and its material is S420 steel. S420 steel has 420 MPa yield and 620 MPa tensile stress. Hyperworks Hypermesh 13.0 was used for modelling data. Four different finite element models were created for optimization. The first FE model was shown in Fig. 2.

The 1st FE model has 7420 elements that are 2D shell and triangle. All elements have 3 connection node and node number is 3911 totally. Boundary conditions are shown in Fig. 2. That is the same for all FE models. Force is 4000 N and its direction is -z. Bracket was fixed with bolt to platform. So

nodes that are around the fixed nodes were fixed for 3 DOFs (z, zy, xz). Those nodes are shown in Fig. 2 with white nodes. After mesh was created in model, some elements can have undesirable sizes. Analysts should check criteria as angle size and aspect ratios. Those criteria can affected models and found out anisotropic problems [5]. Recommended criteria were shown in Fig. 3 [6].

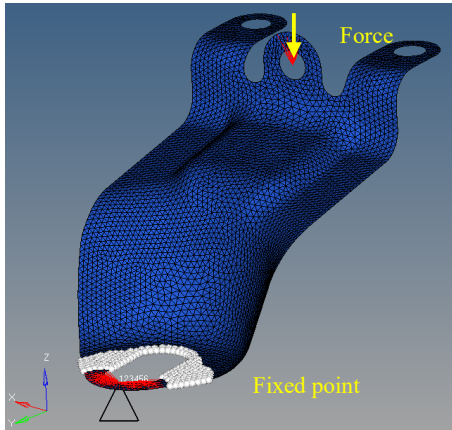


Fig. 2 First FE Model (Shell Tria Element)

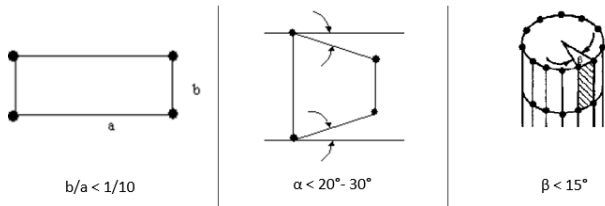


Fig. 3 Recommend Criteria

The 2nd FE model consists of 2D shell and quad elements. This model was shown in Fig. 4.

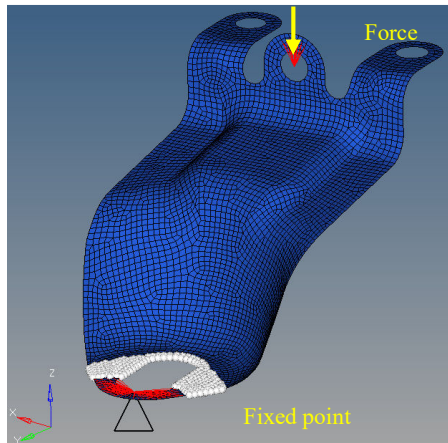


Fig. 4 Second FE Model (Shell Quad Element)

The 2nd FE model contains 4089 nodes and 3907 elements. This model has more nodes and less elements because it is consist of quad elements. Average edge of element is the same with 1st FE model.

There are 2D shell elements in the 1st and the 2nd models. But 3rd and 4th FE models contain solid elements. The 3rd FE model was shown in Fig. 5.

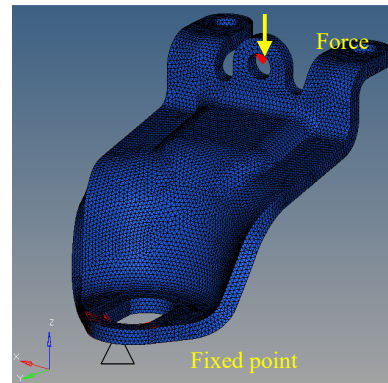


Fig. 5 Third FE Model (Solid Tetra Element)

The 3rd FE model contains 21580 nodes and 91025 elements. Those values are much more than the 1st and the 2nd FE model. Shell models are created with midsurfaces. Midsurface is average surface for sheet metal. Firstly midsurface are created from product cad data and then mesh is created on midsurface. But mesh creating process is different for solid FE models. Product volume is full with elements completely for those models. So element's numbers of solid FE models are much more than shell FE models. Boundary conditions are the same with other models as said before. The 3rd FE model contains tetra elements. All elements have 4 nodes. And lastly the 4th FE model was shown in Fig. 6. The 4th FE model contains 3D solid quad elements and it has 15642 elements and 20462 nodes.

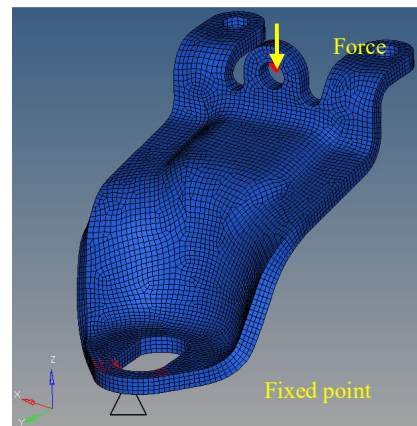


Fig. 6 Fourth FE Model (Solid Hex Element)

Four different analyses was solved with those models. Tests were done with real product for controlling to analysis results (Fig. 7).

Test and analysis result is shown in next section. The 3rd FE model that has maximum elements number, was taken maximum analysis time. Maximum load that applied in analyses is 4000 N but this value is higher in tests. Maximum

4000 N that is in load-stroke curve of test results was taken in consideration.

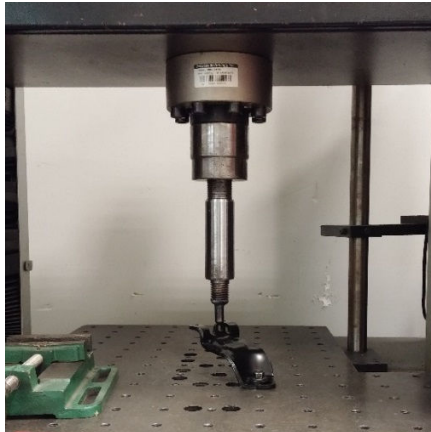


Fig. 7 Product Test

III. RESULTS

Firstly, analysis result was shown in Fig. 8 for FE model that contains 2D tria elements.

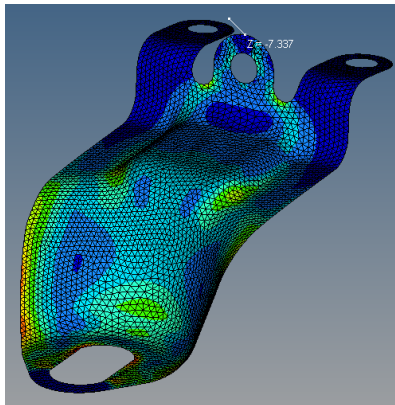


Fig. 8 First FE Model Analysis Result

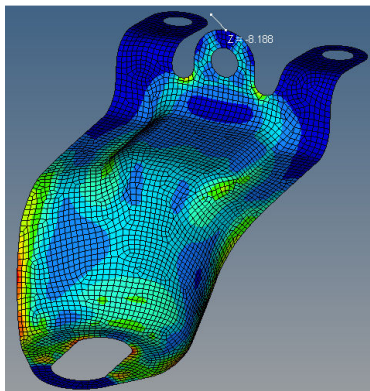


Fig. 9 Second FE Model Analysis Result

According to result, displacement of bracket is 7.33 mm on $-z$ direction. Resultant displacement value was not considered because analysis result was compared with displacement of testing machine on $-z$ direction.

The 2nd analysis was solved with 2D quad elements. Result was shown in Fig. 9. According to result, displacement of bracket is 8.18 mm on $-z$ direction.

3rd and 4th analyses were solved with solid elements. 3th analysis was shown in Fig. 10.

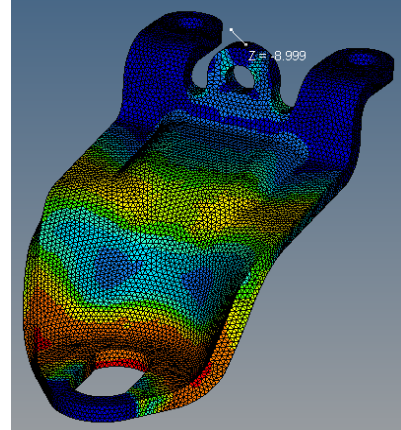


Fig. 10 Third FE Model Analysis Result

In this analysis 3D solid tetra elements were used. According to analysis result, displacement of bracket is 8.99 mm on $-z$ direction. Analysis time of this FE model is maximum between other analyses. After this analysis, 4th FE model that has 3D solid hex elements was solved lastly. The 4th analysis result that was solved was shown in Fig. 11.

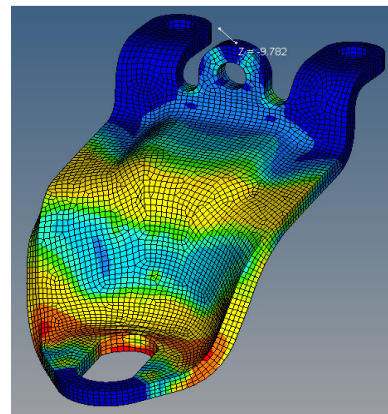


Fig. 11 Fourth FE Model Analysis Result

According to 4th analysis result, displacement of bracket is 9.78 mm on $-z$ direction. In this way analyses were completed. After analyses, 2 tests were done with real products. Test results were shown in Fig. 12. Both of results are close each other. So results are confidential.

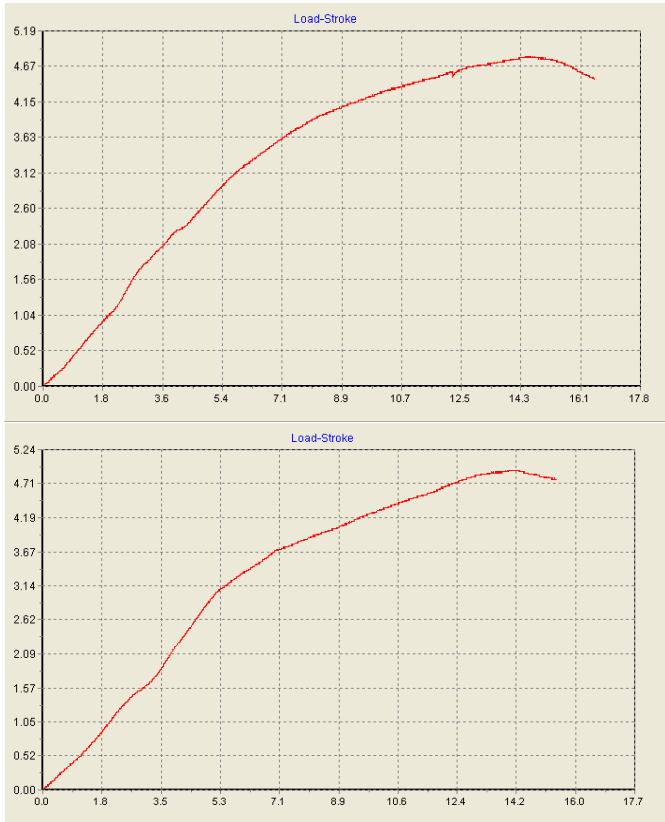


Fig. 12 Test Results

Finally all analysis and test results were shown in Tables I and II.

TABLE I
 ANALYSIS RESULTS

Displacement Results for 4000 N (mm, -z direction)			
1 st FE Model	2 nd FE Model	3 rd FE Model	4 th FE Model
2D Shell Tria	2D Shell Quad	3D Solid Tetra	3D Solid Hex
7,337 mm	8,188 mm	8,999 mm	9,782 mm

TABLE II
 TEST RESULTS

First Test		Second Test	
Load (N)	Stroke (mm)	Load (N)	Stroke (mm)
4000	8,50	4000	8,62

IV. DISCUSSION

According to test results, average real displacement value is 8.56 mm. 2nd FE models result is the closest analysis result to real displacement value. Displacement value of 2nd FE model is 8,188 mm and it contains 2D shell quad elements. On the other hand 4th FE models result is the furthest analysis result to real value. 3D elements have bigger deformation than 2D elements. Difference between 2D and 3D elements deformation is about degrees of freedom's elements and geometry. Elements are meshed on midsurface for 2D FE models but volume is full with elements for 3D FE models. However, it should not be forgotten that product that was tested and analyzed, is sheet metal. So those results are valid for this geometry and boundary conditions.

V. CONCLUSION

It is understood that 2D shell quad elements are the best choice for sheet metal's finite element model according to the results. As well as models that contain 2D elements, have less element. Accordingly those models have less analysis time. More accurate results are taken in less time with 2D shell quad elements for sheet metals.

ACKNOWLEDGMENT

The authors gratefully acknowledge the support of Turkish Technology to Rollmech Automotive San. ve Tic. A.S.

REFERENCES

- [1] S. Piskin, 'Hesaplama Bilim ve Mühendislikte Özel Konular: Mesh Üretimi', ITU, 2002.
- [2] R. L. Courant, 'Variational Methods for the Solution of Problems of Equilibrium and Vibration', Bulletin of the American Mathematical Society, 49, 1943, pp. 1-23.
- [3] F. Williamson, 'Richard Courant and the Finite Element Method: A Further Look', Historia Mathematica 7, 1980, 369-378.
- [4] Adam J. Sadowski, J. Michael Rotter, 'Solid or shell finite elements to model thick cylindrical tubes and shells under global bending', International Journal of Mechanical Sciences, 74, 2013, 143-153
- [5] H. Sun, D. L. Darmofal, R. Haimes, 'Journal of Computational Physics', 231, 2012, 541-557.
- [6] I. Cayiroglu, 'Bilgisayar Destekli Tasarım ve Analiz' lesson notes, <http://www.ibrahimcayiroglu.com>.