International Journal of Mechanical Engineering and Technology (IJMET)
Volume 9, Issue 9, September 2018, pp. 1348–1355, Article ID: IJMET_09_09_148
Available online at http://www.iaeme.com/ijmet/issues.asp?JType=IJMET&VType=9&IType=9
ISSN Print: 0976-6340 and ISSN Online: 0976-6359

FEM ANALYSIS OF THE EFFECT MESH SIZE FOR I-SECTION BEAM

Al-Hadrayi Ziadoon M.R, Marwah Mohammed Abdulridha, Ekhlas Hasan Kadhim
Department of Materials Engineering,
Faculty of Engineering, Kufa University, Iraq.

ABSTRACT

The critical problem in the finite element analysis is mesh size. The mesh size is relating to number and accuracy of what mesh required for the element meshing. This paper was showed the effect of mesh sizes on the accuracy of the results and indicated the convenient mesh size which provided in finite element analysis. Models were carried out by ABAQUS. The material is aluminum alloys 6061 I-section beam. The type of loading is uniformly distributing on the top surface of the beam. All dimensions and mechanical properties have been illustrated in this paper. Numerical models were investigated different mesh sizes of element. This study has been showed the effect of different mish sizes on the result such as stress concentration, displacement, strain energy and maximum stress-mesh sizes. The models have been developed by using ABAQUS and have been compared with others. In this study, explore the relation between the error percentage and mesh sizes for Vonmises stress and the deflection.

Key words: Finite element, mesh size, stress, strain energy, deflection.

Cite this Article: Al-Hadrayi Ziadoon M.R, Marwah Mohammed Abdulridha, Ekhlas Hasan Kadhim, Fem Analysis of the Effect Mesh Size for I-Section Beam, *International Journal of Mechanical Engineering and Technology* 9(9), 2018, pp. 1348–1355.
http://www.iaeme.com/IJMET/issues.asp?JType=IJMET&VType=9&IType=9

1. INTRODUCTION

Many researchers have topics about finite element analysis and the effect of mesh size has a role in these studies. The interesting in this subjects have increasing over recent years because of the development of advanced capability of simulation especially finite element analysis. Both the statistical and analytical sides of scaling sizes may effect on the structural and the failure theories of the material [1-3]. The structures are building as discontinuities are very important design especially in the aerospace applications and ships etc. when the cross-section in the structure changes suddenly, the best way to analysis the problems and the behavior the system using the finite element analysis, takes the main role in this side in the recent years. The large-scale simulations have a common process for analysis the responding the structure because of the impact and grounding, such as [4] to remove the elements or to allow the elements are separation [5]. For many years, the researchers are founded the strain to failure value is dependent on the mesh and trials to find their relations which were made, however the results are based on empirical relations. Such as the Offshore Structures Congress and the international ship has reviewed the strain rupture on ship's grounding and collision [6]. It is necessary to study the relation between the strain rupture and mesh size. Therefore, this paper deals with effect of mesh size element on the accuracy of the results.

2. MODEL OF CASE-STUDY

The model of case study is made up of aluminum alloy 6061-T6 due to the applications of this alloy. The cross-section of the beam is type I-section according to the Aluminum Association Structural "I" Beam [7]. The reason to select this cross-section type because of many applications especially in the wings of aircraft and the steel structure of buildings. The model dimensions are illustrated in the figure (1) and the table (1). The mechanical properties are introduced in the table (2) and the load which used of 100 KN on top surface.

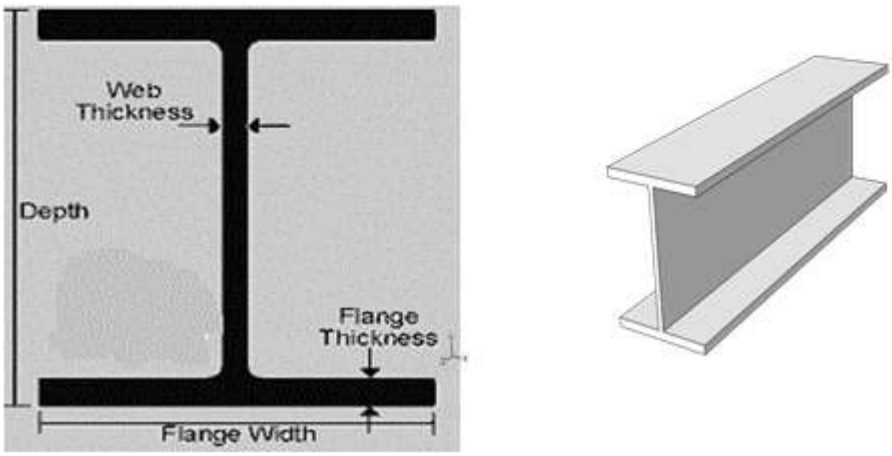


Figure 1 sketch of I-section Beam

Table 1 All dimensions of beam

Depth mm	Flange thickness mm	Web thickness mm	Flange width mm	Beam Length mm
340	20	13	250	1000

Table 2 Mechanical Properties of aluminum alloy 6061-T6

Property, Unit	**Value**
Elastic Modulus, GPa	68.9
Poisson's ratio	0.32
Density, g/cc	2.7

3. FINITE ELEMENT ANALYSIS

3.1. FEM Model

All models are implemented into software ABAQUS. Different meshes sizes of elements have used as shown in the figure (2) which illustrated the sketch of different meshes sizes have used. The meshes sizes are (10, 50, 100, 200, 250 and 500 mm) as shown in the figure (2). The element library is standard and the geometry is linear with family 3D stress.

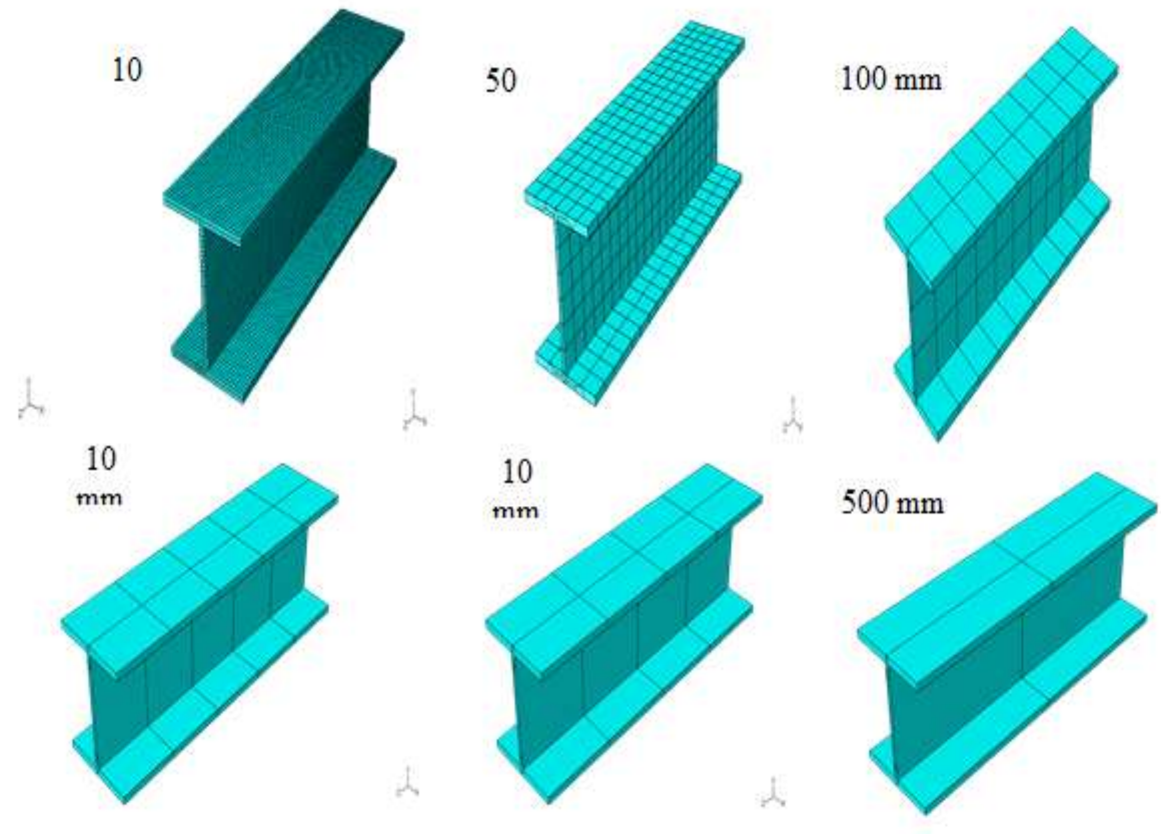


Figure 2 Different meshes size for elements

3.2. Load & Boundary Conditions

The figure (3) presents the load that used on the top surface of the model as illustrated in the figure. The boundary conditions have been arrested two sides of edges of the model that making the (U1, U2, U3) is zero.

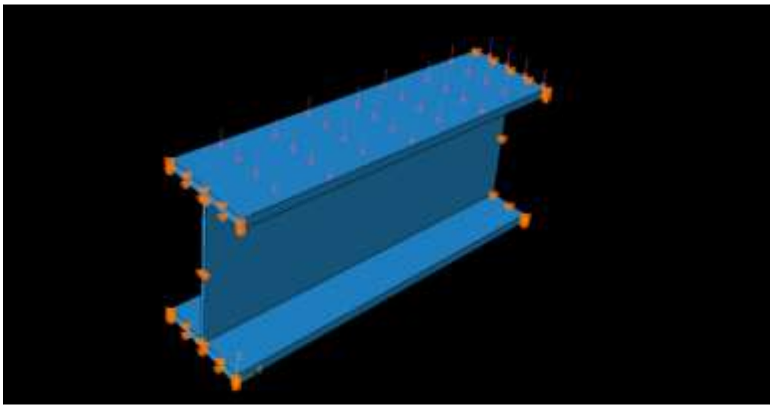


Figure 3 Load & B.C

4. RESULTS AND DISCUSSIONS

The figure (4) has introduced the stress distribution for each model with different size of element to show the effect of mesh size. The models have developed by ABAQUS. Figure (4 a) presents the mesh size 10 mm which consider is best result and compared it with the rest.

At this model, the maximum stress is 6.66 MPa. Also, the displacement has been presented in the figure (5) and calculated for each model. Figure (5 a) show the displacement and the maximum deflection is $1.128 * 10^{-3}$ mm which consider the best result also when it compared it with others. Static analysis for each model has been calculated and show the behavior of models with different size of element as shown in the figure (6 a) and the error percentage of stress and deflection which introduced in the figure (6 b).

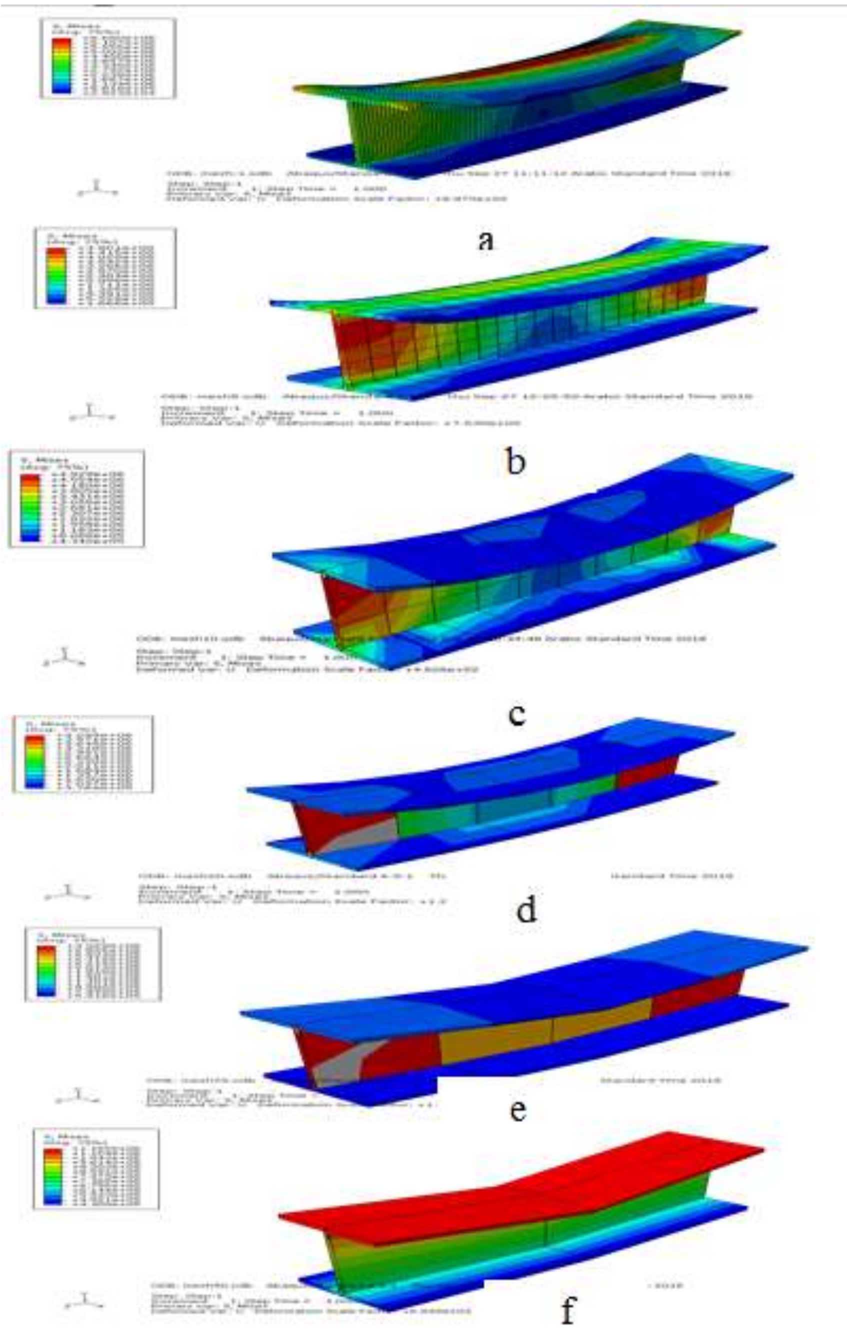
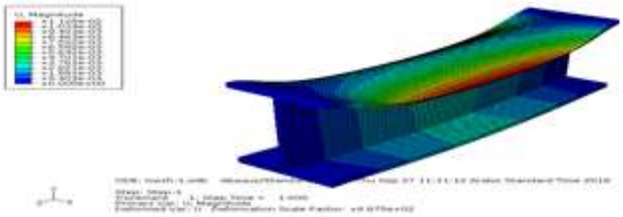
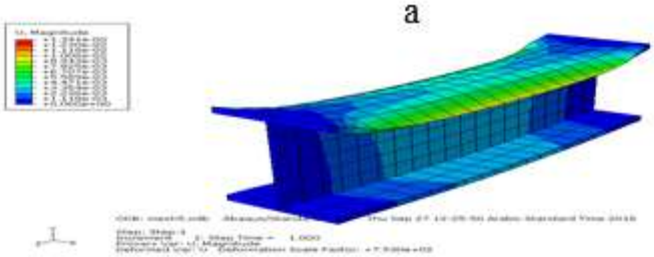


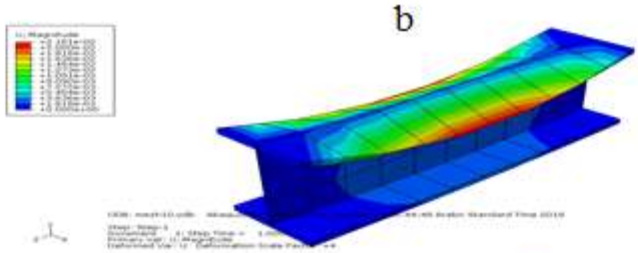
Figure 4 stress distribution for different models according to element size (a) 10 mm (b) 50 mm (c) 100 mm (d) 200 mm (e) 250 mm (f) 500 mm



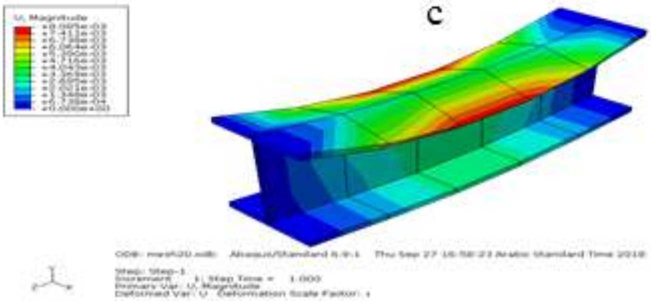
a



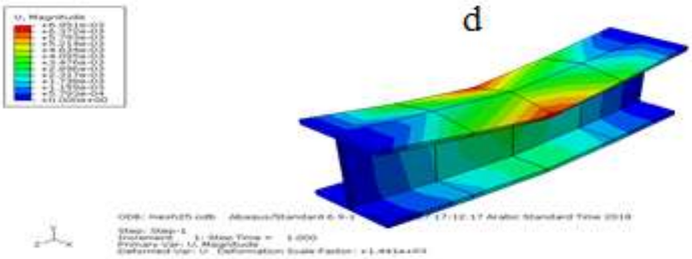
b



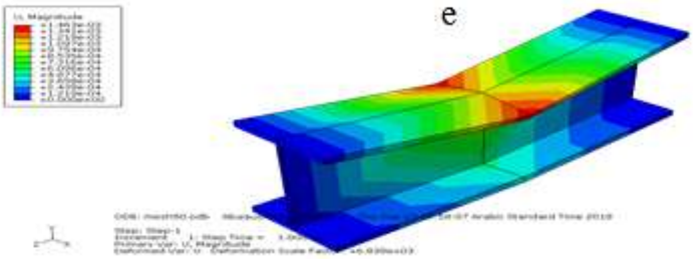
c



d



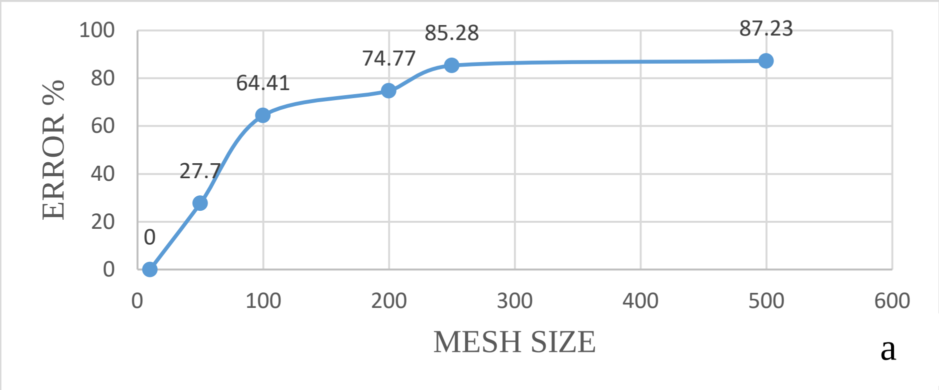
e



f

Figure 5 displacement for different models according to element size (a) 10 mm (b) 50 mm (c) 100 mm (d) 200 mm (e) 250 mm (f) 500 mm

The figure (7) presented the behavior of models at the elements which calculated the stress which located along the path of top edge. While figure (8) presented the deflection behavior of the same path based on the mesh size.



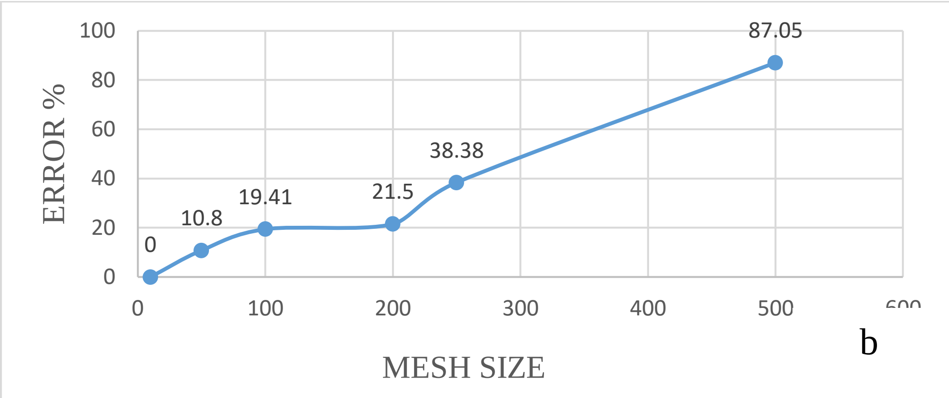


Figure 6 Error percentage of (a) stress and (b) deflection

It has noted that the error percentage started from 0 to 87.23 % increasing when the size of element is increased in the both cases of stress and the deflection.

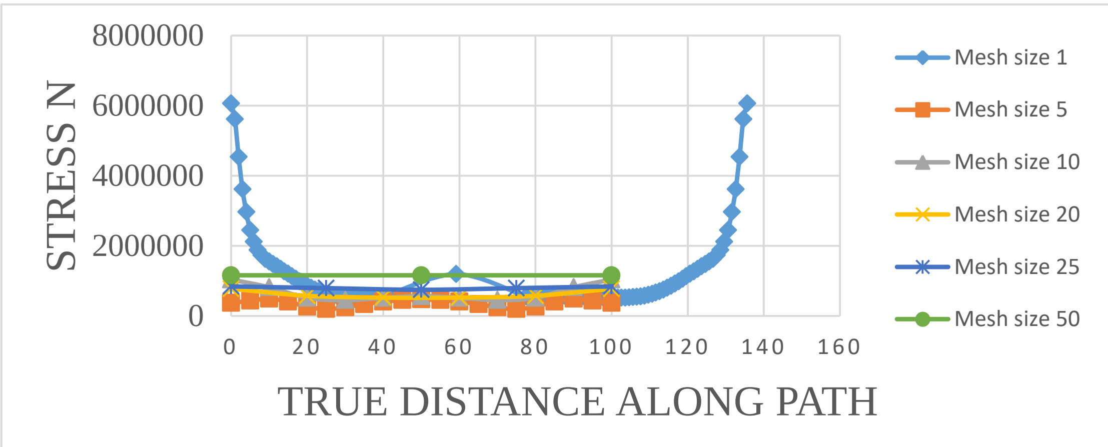


Figure 7 Stress distribution along top edge path

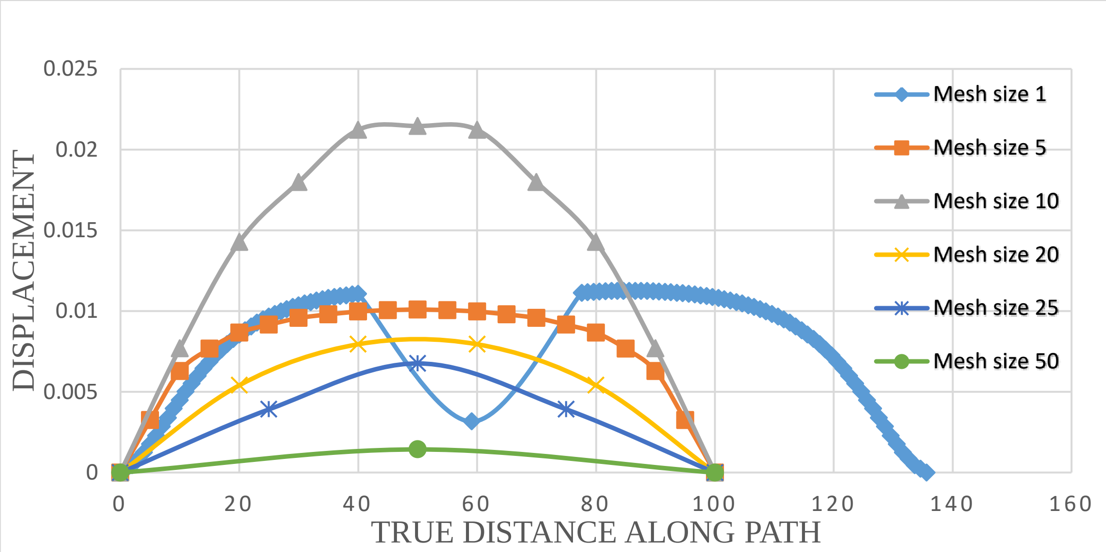


Figure 8 deflection along top edge path

It has noted that at element size of 500 mm the line became as straight. While the element size of 10 mm gave good behavior for model and the length of beam became 1350 mm. A good accuracy result got from the model at element size of 10 mm.

The figure (9) has been presented the strain energy in the true distance along path at the top edge. It has noted that when the element size of 10 mm is better than behavior of elements sizes of 50, 100, 200, 250, 500 mm.

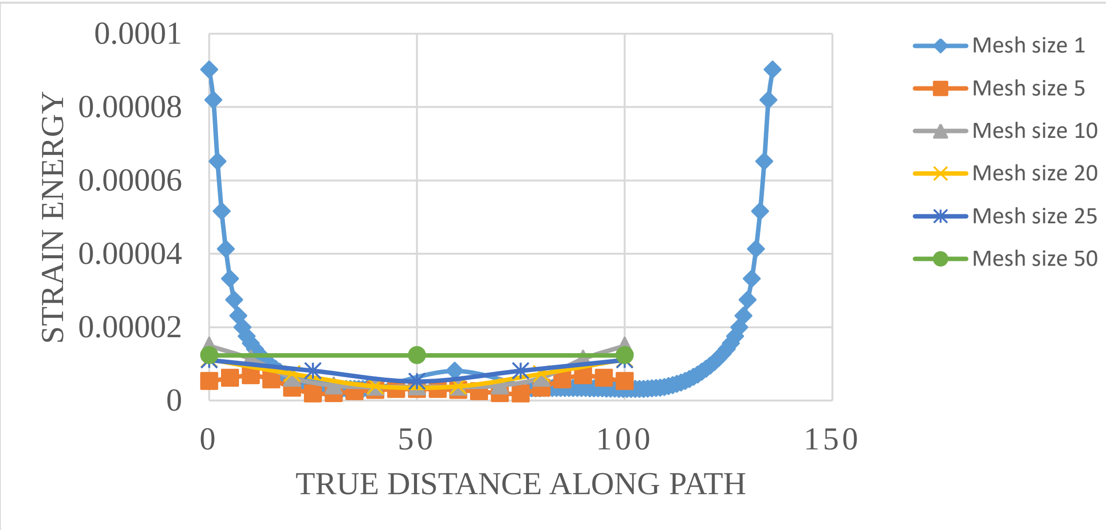


Figure 9 Strain energy along top edge path

5. CONCLUSIONS

In this study, elements sizes have been investigated in the finite element models and the simulations which carried out by using ABAQUS. The results were founded the effect of mesh size on the accuracy of the results such as stress distribution, displacement or deflection and the strain energy. Through the results, static analysis and the pressure loading case which assumed the steady state loading on the top surface of model, it has been found the effect of element size on the results. It was very clear that effecting. When the element size of 10 mm gave good results and better than others. The percentage of errors were increasing when element size is increasing. Also ABAQUS provide any size of elements less than 10 mm can generate models to get the best results.

REFERENCES

[1] Bažant, P. Z., 1997. Scaling of Structural Failure. Appl. Mech. Rec. Vol. 50, No.10, 593-620.

[2] Bažant, Z.P., 2000. Size effect, International Journal of Solids and Structures 37, 69-80.

[3] Bažant, Z.P., Guo, Z., 2002. Size effect and asymptotic matching approximations in strain-gradient theories of micro-scale plasticity, International Journal of Solids and Structures 39, 5633-5657.

[4] Li, Y.N., Gao, Z. and Gu, Y., 2002. The Benchmark Study of the Tensile Coupon Test and Model Test of Ship Grounding. Journal of Ship Engineering 6, 13-17. (in Chinese)

[5] Simonsen, B. C., Tornqvist, R., 2004. Experimental and numerical modeling of ductile crack propagation in largescale shell structures, Marine Structures 17, 1-27.

[6] Wang, G. 2006. ISSC V.1 Committee Collision and Grounding, ISSC, Southampton, UK

[7] http://www.metalreference.com/INFO_Aluminum.html