EAN 2020
58th conference on experimental stress analysis

Numerical and Experimental Analysis of C/PPS Y-Shaped Profile

PADOVEC Z.[1,a], ZÁMEČNÍKOVÁ T.[1,b], SEDLÁČEK R.[1,c]

[1]Deparment of Mechanics, Biomechanics and Mechatronics, Faculty of Mechanical Engineering; Czech Technical University in Prague; Technická 4, 160 00 Praha, Czech Republic

[a]zdenek.padovec@fs.cvut.cz, [b]tereza.zamecnikova@fs.cvut.cz, [c]radek.sedlacek@fs.cvut.cz

Keywords: Composite structure, FEM, Experiment

Abstract. Presented paper deals with testing and its FE simulation of Y shaped composite profile designed for joining of aircraft construction. This type of profile was derived and designed according to previous design, testing and simulation of T profile manufactured from same material and with the same lay-up. Results from both experiments were compared with each other with conclusion that new Y shape design can withstand higher loading than T profile.

Introduction

Main goal of this work was testing, and simulation of Y-shaped profile manufactured from C/PPS (carbon fiber, polyphenylene sulphide matrix, fabric with 5H satin weave 0.33 mm thick) composite which will be used for connection of aircraft keel beam with the ribs. Profile was developed in two construction versions – T (which was tested and analysed in [1]) and Y. Results from both tested profiles were evaluated and compared with each other. Both profiles were manufactured by thermoforming process from three plates with $[[(0/90)/(\pm 45)]_2]_s$ lay-up.

Experiment. Experiment was done on TIRA 2300 universal testing machine with loading speed of 2 mm/min (respectively 0.5 mm/min). Tensile load was realized through the screws in the web of the element jointed with the jaws of the machine. Relationship between loading force and displacement for three tested Y and five tested T specimens can be seen in Fig. 1.

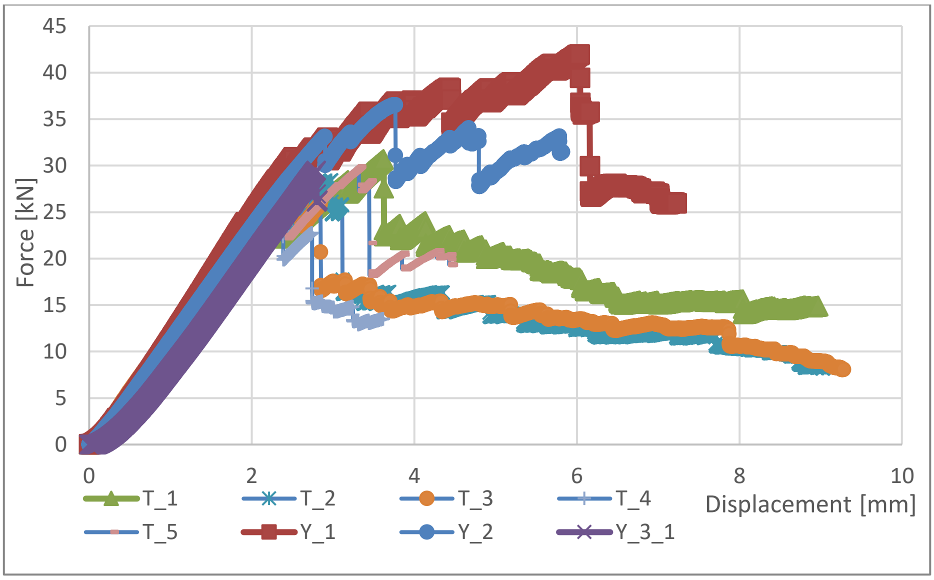

Fig. 1 Relationship between loading force and displacement for tested specimens

First two Y specimens were tested until total failure, third one was tested until first failure occurred. Then the specimen was removed and put into CT scan to find out the area of the failure (see Fig. 2). This failure occurs under the loading force 29.5 kN.

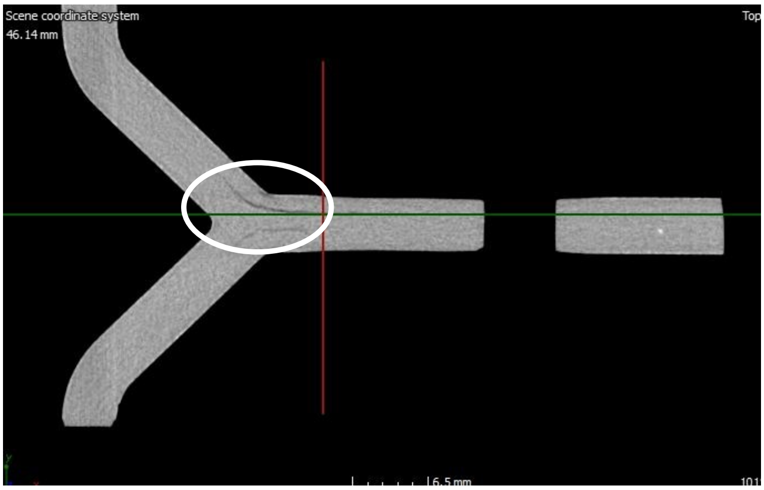

Fig. 2 Failure area evaluated from CT scan

FEM. Simulation of the experiment was done in similar way as simulation of T profile with the use of Abaqus software [1]. The main goal of the simulation was determination of through thickness stress direction in the failed area taken from CT scan. That´s why layered solid elements were used instead of shell or continuum shell elements which can´t be used because of plane stress assumption. Loading was distributed through reference points in the web holes coupled with their area. Material parameters (taken from technical sheet for certified material used in aircraft industry [3]) of each layer were entered as engineering constants (see Table 1), area of pure matrix was modeled as isotropic material (see Table 2). Material orientation can be seen in Fig. 3.

Table 1: Material parameters for composite layer [3]

$E_1 = E_2$ [MPa]	E_3 [MPa]	G_{12} [MPa]	$G_{23} = G_{13}$ [MPa]	ν_{12} [-]	$\nu_{23} = \nu_{13}$ [-]
58 000	10 000	4 100	3 500	0.046	0.33

Table 2: Material parameters for pure matrix

E [MPa]	ν [-]
3 800	0.33

FE model of the experiment, stress field on whole profile and stress field in detail in the failed area can be seen in Figures 4 – 6.

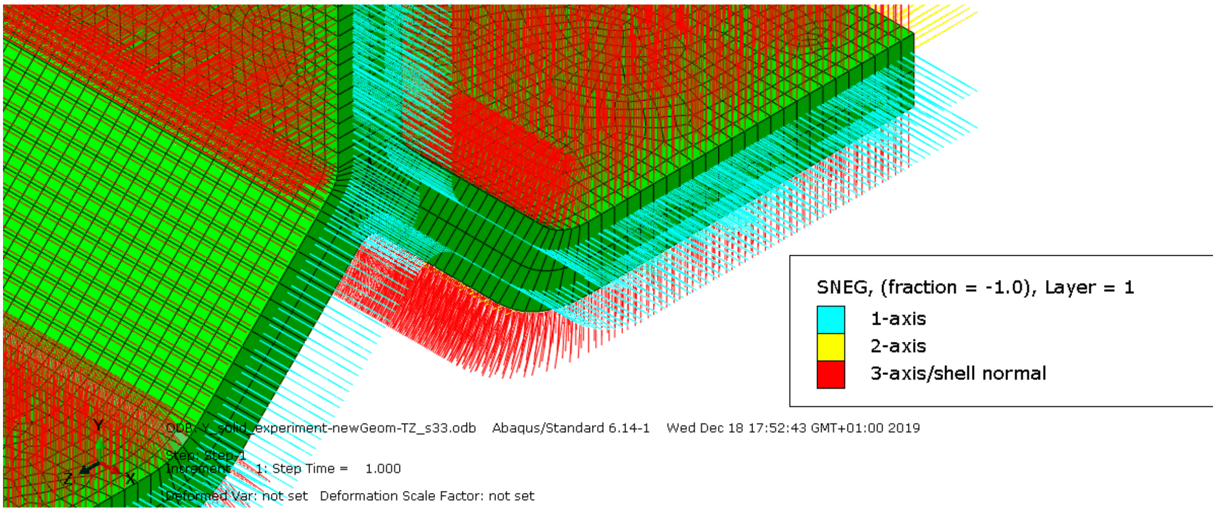

Fig. 3 Material orientation in the beam

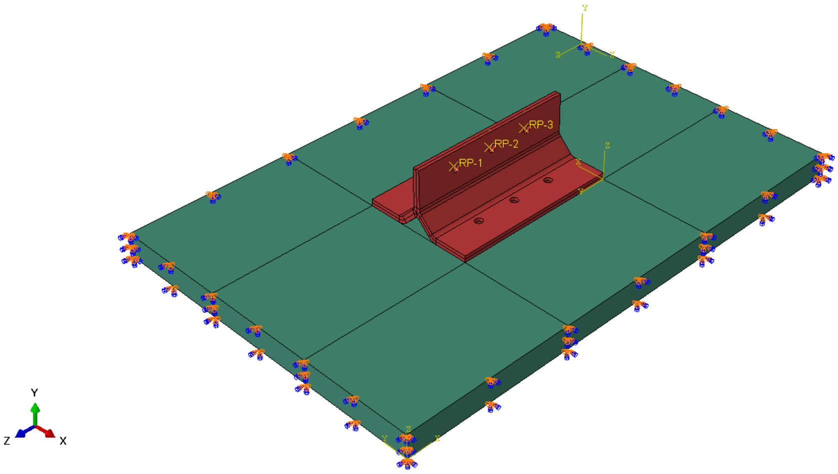

Fig. 4 FE model of the experiment

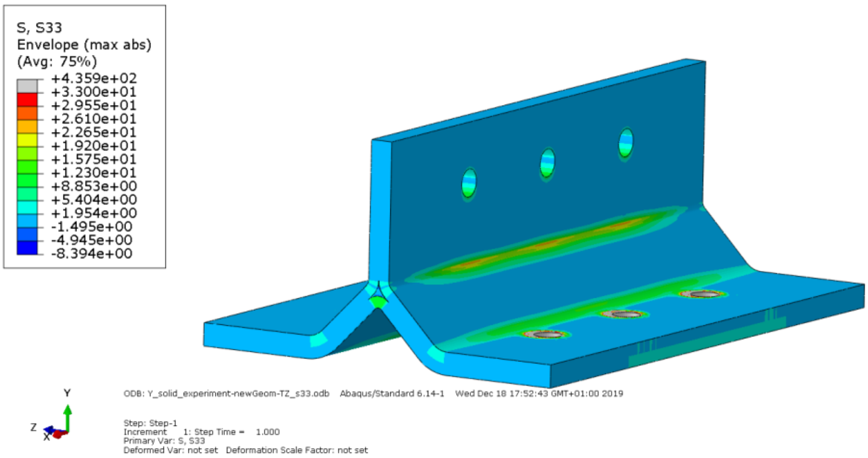

Fig. 5 S33 stress field in whole profile

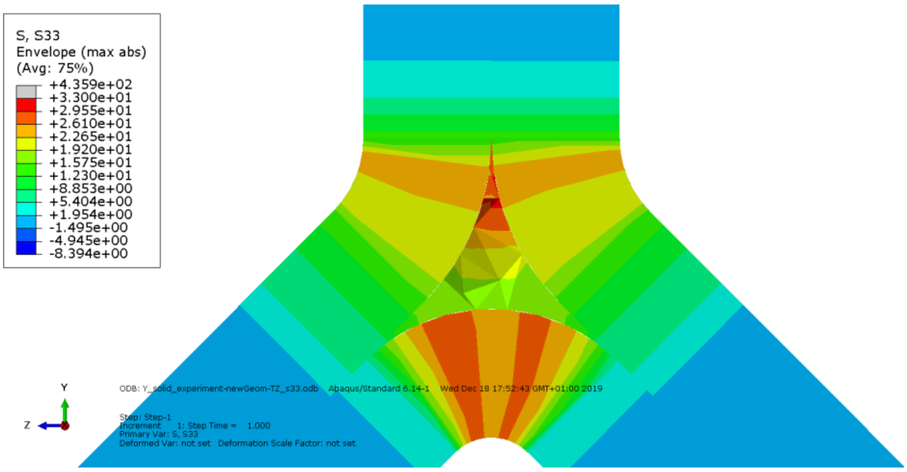

Fig. 6 S33 stress field in the failed area

Conclusions

If we compare maximal value of stress from FE simulation which is 33 MPa and presumed allowed value from stress taken from material sheet [2] which is 39 MPa we see that the agreement between simulation and experiment is very good. Failure theories for composites implemented in Abaqus (maximum stress, maximum strain, Tsai-Hill, Tsai-Wu and Azzi-Tsai-Hill) can´t be used because the shell/continuum shell elements which assume plane stress condition (S33 = 0) were not used in this analysis. Also, the area of the failure from CT scan compared with the maximal stress area from FE simulation shows good agreement.

If we compare results from the experiment of T profile and Y profile (Figure 1) we can see that the first decrease of the loading force (which is crucial for fatigue) in tested T profiles occurs between 14 and 16 kN. In tested Y profiles this decrease occurs between 24 and 34 kN which means circa 71 – 112 % improvement compared to T profile results.

Acknowledgements

This work has been supported by project FV30033 of the Ministry of Industry and Trade of the Czech Republic and by the Grant Agency of the Czech Technical University in Prague, under grant No. SGS18/175/OHK2/3T/12.

References

[1] Z. Padovec, R. Sedláček, T. Zámečníková, Testing of C/PPS Element for Aircraft, in: EAN 2019 57th conference on experimental stress analysis CONFERENCE PROCEEDINGS, Luhačovice, Czech Republic, 2019, pp. 361-364.

[2] TenCate Cetex® TC1100 PPS Resin System, Product Data.

[3] AIMS05-09-002, Airbus Material Specification Carbon Fabric, 285 g/m^2 fiber area mass with 43% PPS Resin or Equivalent resin Material Specification.