Numerical and experimental analysis of compressor's jet engine blade joint including the model parameterization

Numeryczno-doświadczalna analiza połączenia zamkowego z uwzględnieniem parametryzacji modelu

ADAM KOZAKIEWICZ
OLGA GRZEJSZCZAK
TOMASZ ŁĄCKI *

DOI: https://doi.org/10.17814/mechanik.2017.7.73

Presented is the strength analysis of three variants of the compressor's jet engine blade joint whose final geometry was created using the ANSYS optimization module. Parameterization of the blade foot and rim geometry has been introduced and a comparative analysis of the selected geometry in terms of strength has been performed. In addition, results from the experimental analysis of the substitute models of blade joint with using the digital image correlation system were presented.
KEYWORDS: stress analysis, parameterization, dovetail joint, optimization, jet engine

Parameterization, that is, introduction of variables to the geometrical model, allows to effectively trace the influence of selected parameters on the work and behavior of a given structure. In addition, it accelerates model calculations and makes changes possible at every design stage. Combined with the optimization process, it enables the development of multiple design variants (with different geometric dimensions and features) and the choice of the best solution according to the optimization criterion. Calculations can be performed automatically, in a relatively short time, which makes the process of design and construction is much more efficient.

The subject of the study is a comparative analysis of three computational variants, based on the modeled and parameterized trapezoidal compressor's jet engine blade joint using ANSYS optimization tools. In the first step, numerical finite element strength analysis was performed to obtain the distribution of stress and displacement at the centrifugal force. At the next stage, the blade-disk connection was subjected to a tensile strength test. The Dantec Q400 digital image correlation system was used to measure the deformation and to visualize the deformation of the actual surface.

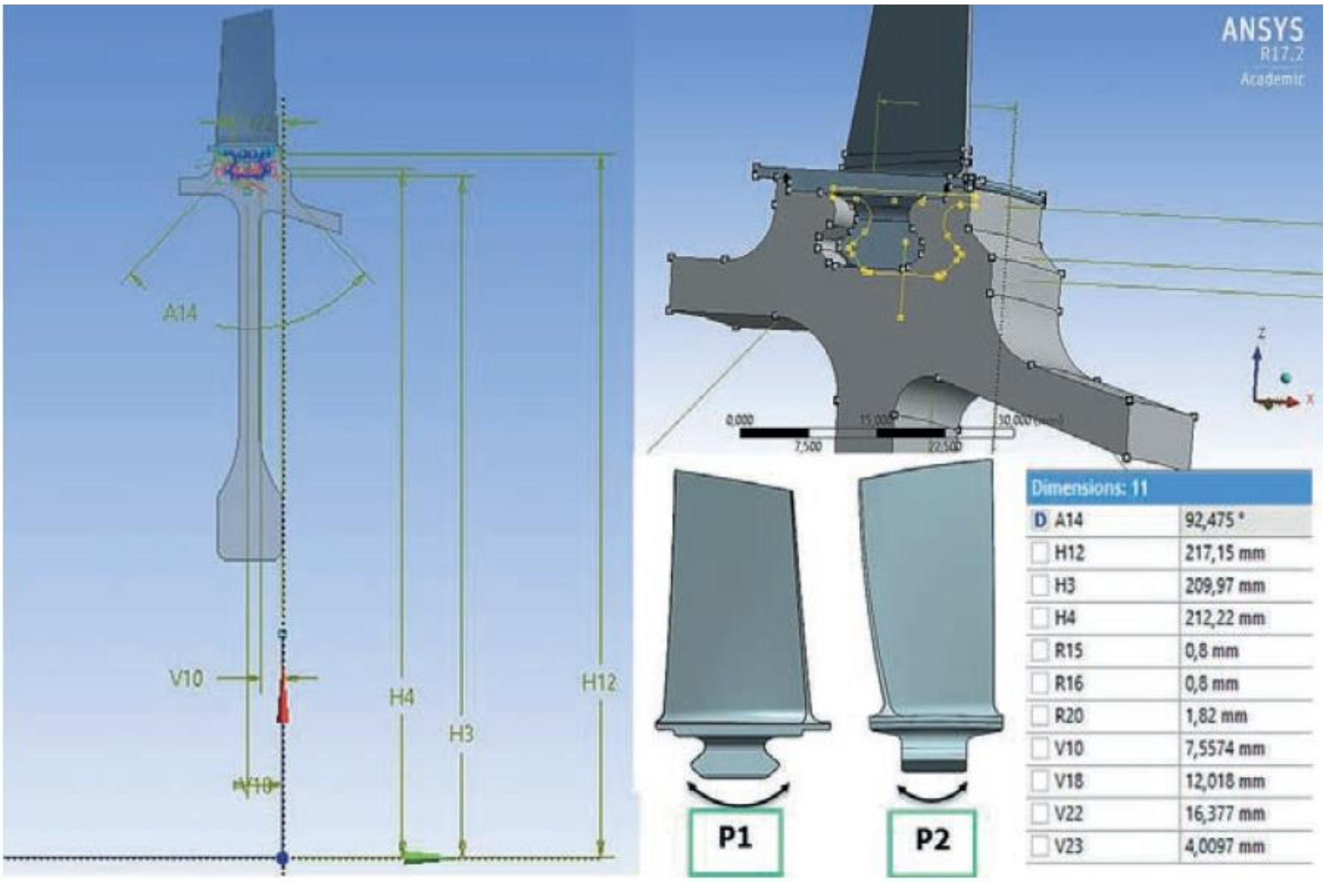


Fig. 1. Geometry of the output model

* Dr hab. inż. Adam Kozakiewicz (adam.kozakiewicz@wat.edu.pl), mgr inż. Olga Grzejszczak (olga.grzejszczak@wat.edu.pl), mgr inż. Tomasz Łącki (tomasz.lacki@wat.edu.pl) – Wydział Mechatroniki i Lotnictwa Wojskowej Akademii Technicznej

Numerical analysis

ANSYS software allows to define both variables and geometric constraints at the geometry creation stage. An effective tool in ANSYS DesignXplorer, which allows you to analyze the variability of an interesting parameter with respect to input parameters, is Response Surface. By specifying the range of variability of the selected parameters, one can allow the program to randomly select several combinations of these parameters or enter values yourself. There are several or more variants of the model with corresponding values of maximum stresses, deformations, etc. The obtained data can be further analyzed using the Response Surface Optimization tool. It uses the selected optimization method and searches and selects the values that best meet the criteria.

The object of the study is trapezoidal peripheral lock which is most commonly used as the connection type of the disc blade in a rotor axial compressors. Output geometry footer has been mapped using reverse engineering methods. CAD model created in the program Siemens NX. For the purposes of parameterization, the geometry of the footer and the disk rim were made directly in an ANSYS (fig. 1).

Of the several geometric parameters for further analysis, the following were chosen:

- parameter P1 – footer thickness defined by angle A14 (fig. 1, left),
- parameter P2 – width of the footer (fig. 1, right).

The effect of these parameters on the stresses and displacements in the lock at the inertia load resulting from the assumed rotational speed was influenced. 30 variants were analyzed. Parameter P1 changed in the range 86÷98°, and the parameter P2 – in the range of 1.26÷1.6°. The results are presented in fig. 2. At the next stage of the analysis, two geometric variants of the test foot (Table I) were selected based on optimization. The first met the criterion of minimum stresses and deformations (MIN), the second – the criterion of maximum stresses and deformations (MAX).

Further numerical analysis and experiments were carried out for the geometrical variants indicated in Table I as W1 and W4. For comparison, the geometry of the output was also examined. The results of strength analysis, including the numerical determination of reduced stress distribution and deformation, are presented in fig. 3 and fig. 4.

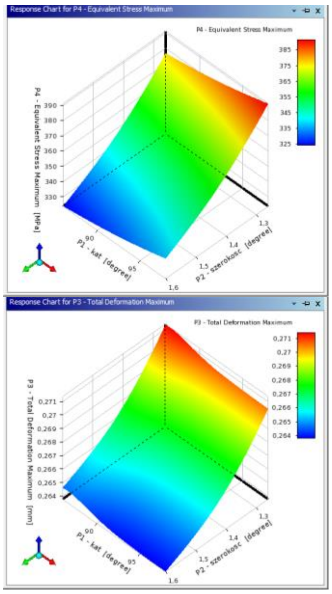


Fig. 2. Influence of parameters P1 and P2 on the distribution of equivalent (von Mises) stress and deformations at rotational speed ω = 927 rad/s

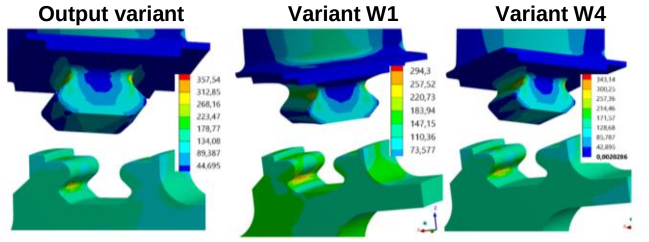


Fig. 3. Equivalent (von Mises) stress distribution, MPa

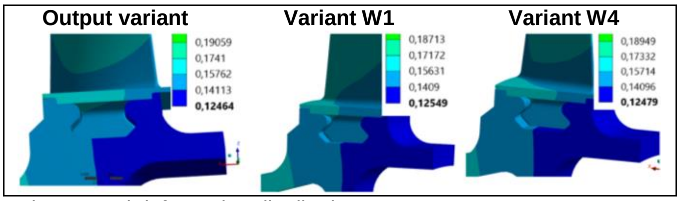


Fig. 4. Total deformation distribution, mm

TABLE I. Geometric variants of the joint

	MIN criterion			MAX criterion		
Parameter	W1	W2	W3	W4	W5	W6
P1, degree	86	95.3	98	98	86.60	93.74
P2, degree	1.6	1.59	1.6	1.26	1.26	1.26
Deformation, mm	0.27	0.26	0.26	0.27	0.27	0.27
Stress, MPa	323.99	334.6	337.15	391.2	377.27	383.69

Experimental analysis

Experimental analysis included a static tensile strain test of trapezoidal locks and strain measurement at a given load, varying from 0.5 to 10 kN. A digital image correlation system was used to determine the deformation areas of the examined object. The principle of this system consists in illuminating and analyzing the intensity of light reflected from the surface of the object in the non-deformed (initial) and deformed state (fig. 5) [1].

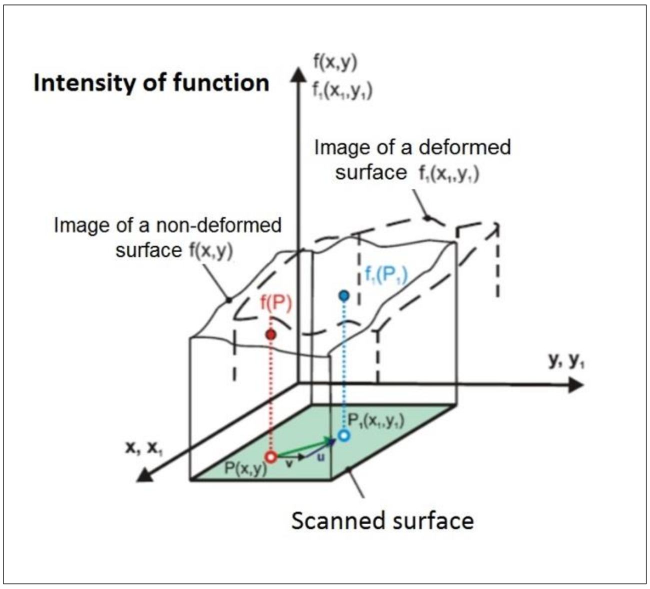
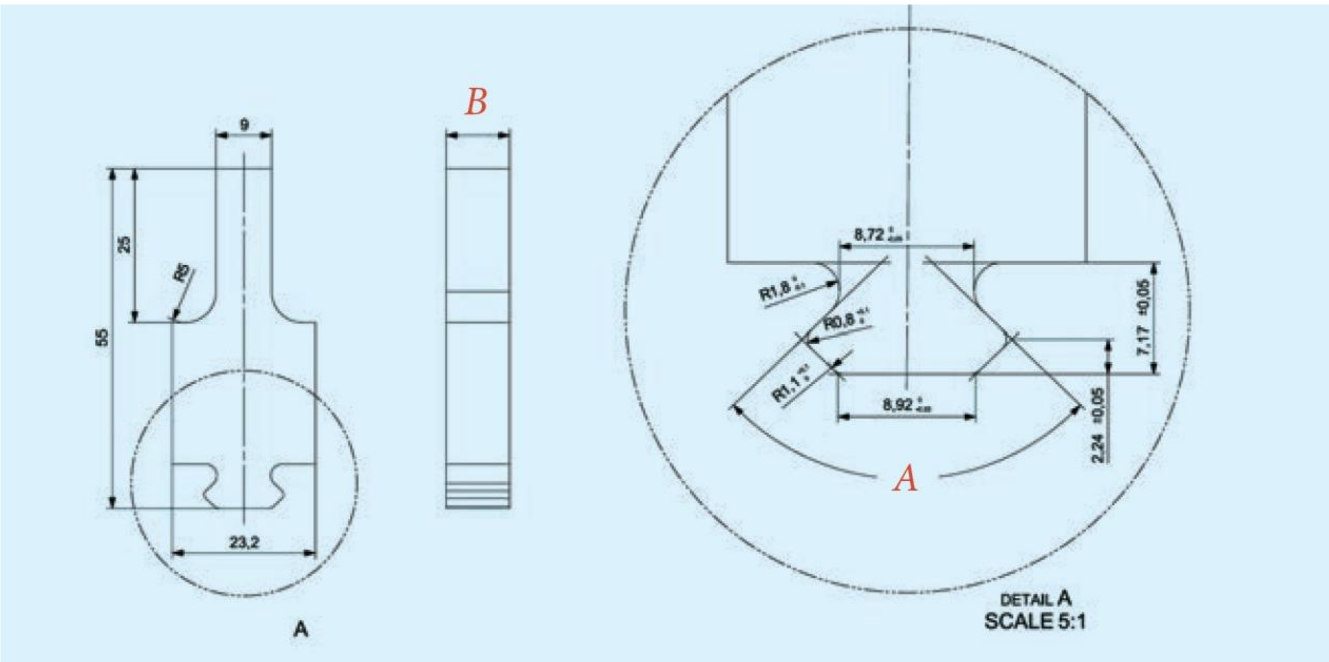


Fig. 5. Analysis scheme of undeformed and deformed surface image [1]

Three models made of low carbon steel 1.7734.4 with a density of 7850 kg/m^3 were subject to analysis. The geometrical dimensions of samples are shown in fig. 6.

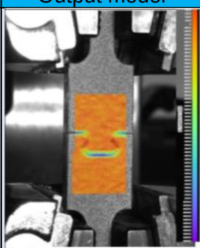
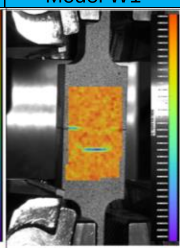
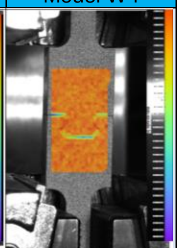
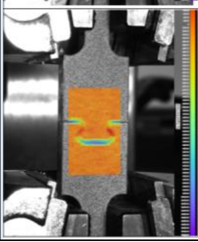
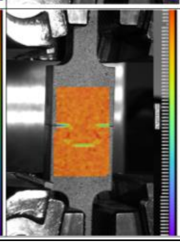
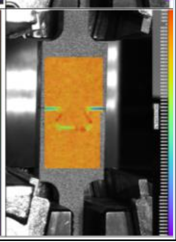


	Output model	Model W1	Model W4
A, °	92.48	86	98
B, mm	10.26	11.73	9.23

Fig. 6. Dimensions of the samples tested

Approximately 60 measurements were made. Table II shows the results of the deformation fields at 5 and 10 kN load.

TABLE II. Sample deformation fields

Force, kN	Output model	Model W1	Model W4
5			
10			

Analyzes have shown that the geometric parameters have a rather large effect on the blade-blade connection strength. In numerical and experimental simulations, comparable stress values were obtained in places of increased concentration. In numerical analysis, with a load of about 5 kN, the maximum stresses on the foot were about 290÷410 MPa. In the experimental analysis, for the analogous load, the deformations obtained correspond to the stresses of 294÷420 MPa, depending on the geometric variant.

Conclusions

Three geometric variants of the trapezoidal blade-to-disk combination of inertial forces were analyzed. Stress and strain distributions are presented for each computational case. The geometry calculation scheme for ANSYS geometry optimization, based on which numerical models were also used in experimental analysis, is discussed. The use of the parametric model and ANSYS tools has resulted in a large number of parameter combinations and a quick analysis of the impact of their changes on the structure. Experience has shown the distribution of deformations on the actual surface of the elements, as well as the location of the increased concentration.

REFERENCES

1. Chu T.C., Ranson W.F., Sutton M.A. Peters W.H. „Application of digital-image-correlation techniques to experimental mechanics”. *Experimental Mechanics.* 25, 3 (1985): pages 232–244.
2. Kachel S. „Metoda wielokryterialnego projektowania bryły samolotu z uwzględnieniem misji”. Rozprawa habilitacyjna. Warszawa: Wojskowa Akademia Techniczna, 2011.
3. Szymczak T., Grzywna P., Kowalewski Z.L. „Nowoczesne metody określania wytrzymałościowych właściwości materiałów konstrukcyjnych”. *Transport Samochodowy.* 1 (2013): pages 79–104.
4. ANSYS 17.1 Workbench Toolbox.
5. www.dantecdynamics.com.