STRESS AND DEFORMATIONS ON AXIAL BLADE TURBINE CALCULATED BY FINITE ELEMENTS METHOD

Dorian NEDELCU
Universitatea "Eftimie Murgu" din Reşiţa, Facultatea de Inginerie, P-ţa Traian Vuia No. 1-4, Reşiţa, +40-0255-219134
E-mail: d.nedelcu@uem.ro, România
Ioan PĂDUREAN
Universitatea "Politehnica" din Timişoara, Facultatea de Mecanică, B-dul Mihai Viteazu No. 1, 1900 Timişoara, tel. +40-256-403681, E-mail: padurean58@yahoo.com, România

Abstract: The axial blade turbine is a very complex geometry, with variable thickness and variable angle from hub to periphery. Usually, near the hub are localized the high values of the stress. Analytical methods offer an approximative calculus of the stress. Experimental methods can be used, but with expensive costs for blade model and technical measurements devices. As an alternative, numerical finite element method can be used. The paper describe step by step the procedure of finite element method used to calculate deformation and stress values for axial blade turbine, using mixed Autodesk Inventor and Cosmos Design Star software.

Keywords: Stress, deformation, blade, axial, turbine, Design Star.

1. Introduction

Cosmos Design Star allows to perform the following types of analyses [1]:
- linear static analysis;
- natural frequencies and mode shape calculations;
- critical buckling load estimation;
- linear and nonlinear steady state and transient heat transfer analysis;
- nonlinear stress analysis.

Cosmos Design Star, based on the Parasolid geometry engine, also supports ACIS and STEP AP203 standards and can directly open several CAD system files. With these capabilities, Cosmos Design Star can analyze parts and assemblies created in almost any CAD system.

Linear static analysis calculates displacements, strains, stresses, and reaction forces under the effect of applied loads. The geometry is created, as solid entity, in Autodesk Inventor software [2], [3].

Required input for linear static analysis are:
- meshed model;
- material properties;
- adequate restraints to prevent the body from rigid body motion;
- at least one of the following types of loading: concentrated forces, pressure, prescribed nonzero displacements, body forces (gravitational and/or centrifugal).

Output of static analysis are: displacement components (translation in X, Z,Y direction and resultant displacement), reaction force (in X, Z,Y direction and resultant reaction force), normal stress σ (in X, Z,Y direction and von Mises stress value), shear stress τ (in XZ, XZ,YZ direction). The Von Mises stress is computed from the six stress components as follows:

$$\sigma_{VM} = \sqrt{\frac{1}{2}\left[(\sigma_X - \sigma_Y)^2 + (\sigma_X - \sigma_Z)^2 + (\sigma_Y - \sigma_Z)^2\right] + 3\left(\tau_{XY}^2 + \tau_{XZ}^2 + \tau_{YZ}^2\right)} \quad (1)$$

Steps to perform static analysis:
- import the model geometry;
- create a static analysis study;
- define a material;

- define adequate restraints;
- define some type of loading;
- mesh the model;
- start the analysis calculus;
- visualize the results.

2. Import the blade geometry

The geometry of the blade is imported from Autodesk Inventor software as STEP, IGS or SAT file, figure 1.
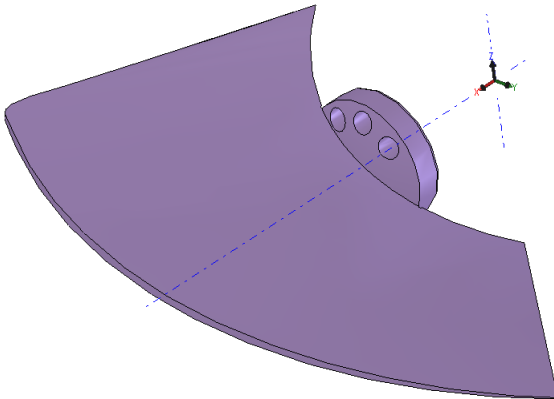
Fig. 1 Imported geometry of the blade

3. Create a static analysis study

When analyzing a part or assembly, it is necessary to investigate its response to various scenarios of service environments and operational conditions. A design study represents a simulation of a "what-if" scenario, completely defined by the type of analysis and related options, material assignments, loads and boundary conditions and mesh. So, it is possible to create a number of studies with different materials, loads, boundary conditions and meshes.

For this analyze will be selected **Static** option for **Analysis Type**, **Solid** option for **Mesh Type** and **FFE** solver.

4. Define a material

For **Static** type analyze it is necessary to define: the Modulus of Elasticity **E** and Poisson's ratio **v**; the density must be defined when considering the effect of gravity and/or centrifugal loading; selecting a material from the COSMOS library, will assign automatically these properties.

5. Define restraints and loads

Cosmos Design Star applies loads and boundary conditions to geometric entities as features that are fully associated with the current geometry.

Restraints must not allow any rigid body motion. The blade will be fixed in the surfaces of the catch holes of the axle pin, figure 2. Fixed restraints will fixes the selected entities in space, so the program will not allow the fixed entities to move in any direction.
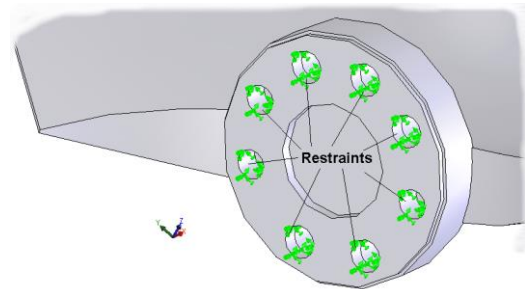
Fig. 2 Restraints applied to the blade

In structural studies to the model can be applied body loads (gravitational and centrifugal forces). Body forces are applied to the whole model. The density of the material must be defined for the program to calculate body forces. For structural studies, can be applied linear accelerations, angular velocities or angular accelerations.

Gravity load is defined by the value of gravitational acceleration, direction of gravity (perpendicular to **Plane4** in this case) and orientation (opposed to Z axis), figure 3.

Centrifugal load is defined by the value of angular velocity (radians or rpm) and rotation axis (**Axis1** in this case), figure 3.
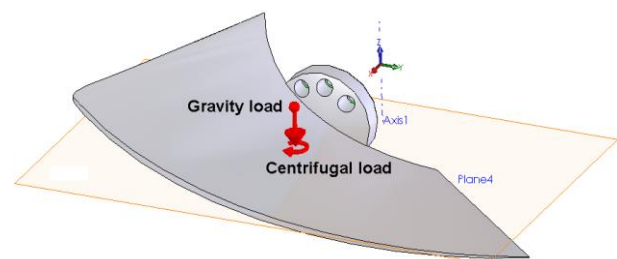
Fig. 3 Gravity and centrifugal loads applied to the blade

Hydrodynamic loads can consist of the following types:
- pressure distribution on the blade; the calculus of the blade pressure distribution require a hydrodynamic software;
- axial and tangential loads on the blade; the values can be obtained experimental from model measurements, figure 4.
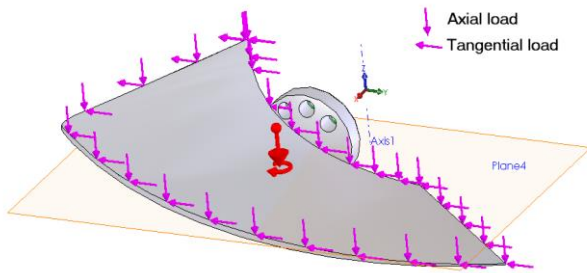
Fig. 4 Axial and tangential loads applied to the blade

6. Mesh the model

Before run a study, the model must be meshed. Each study can have a different mesh. Finite element analysis provides a reliable numerical technique for analyzing engineering designs. The program subdivides the geometric model into small pieces of simple shapes (elements) connected at common points (nodes). Finite element analysis programs look at the model as a network of discrete interconnected elements and assumes that the behaviour of each element varies in particular known fashions for various conditions. The finite element method predicts the behaviour of the model by manipulating the information obtained from all the elements making up the model.

Because the model is a solid entity, tetrahedral solid elements will be used for meshing. When meshing a part or an assembly in a study created with **Solid** mesh type, it is possible to select one of two options:

▪ draft quality mesh: the automatic mesher generates linear tetrahedral solid elements (also called first-order or lower-order elements); a linear tetrahedral element is defined by 4 corner nodes connected by 6 straight edges.

▪ high quality mesh: the automatic mesher generates parabolic tetrahedral solid elements (also called second-order or higher-order elements); a parabolic tetrahedral element is defined by 4 corner nodes, 6 mid-side nodes, and 6 edges.

For the same mesh density (same number of elements and nodes), parabolic elements yield better results than linear elements because: represent curved boundaries more accurately and produce better mathematical approximations. However, parabolic elements require greater computational resources than linear elements.

Each node in a solid element has 3 degrees of freedom which are the translations in three orthogonal directions.

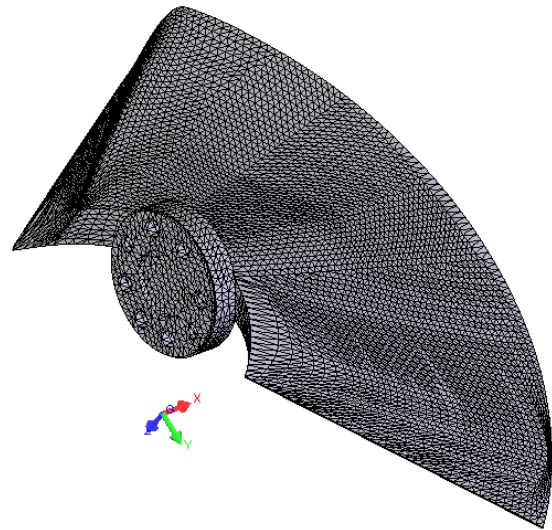The mesh of the blade is presented in figure 5.

Fig. 5 The mesh of the blade

7. Run the analyse study

After completely defining a study, the results can be obtained by running of the analyse study. Running a study calculates the results based on the geometry, material, loads and boundary conditions, and mesh. After the program finishes analyzing the study, there is possible to visualize the results.

8. Visualize the results

After running the analysis Cosmos Design Star automatically creates default result plots for each study type.

Results can be viewed as numerical values and graphically at a specified location, on selected geometry, for model, controls sections and iso plots.

The Von Mises distribution is presented in figure 6. The maximum value is obtained near the intersection point between hub sphere and axle pin cylinder.
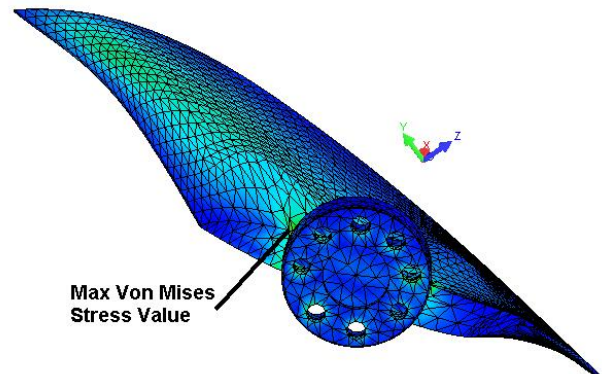

Fig. 6 The Von Mises distribution

The displacement distribution is presented in figure 7 and 8. The maximum value is obtained near the intersection point between periphery sphere and trailing edge surface.
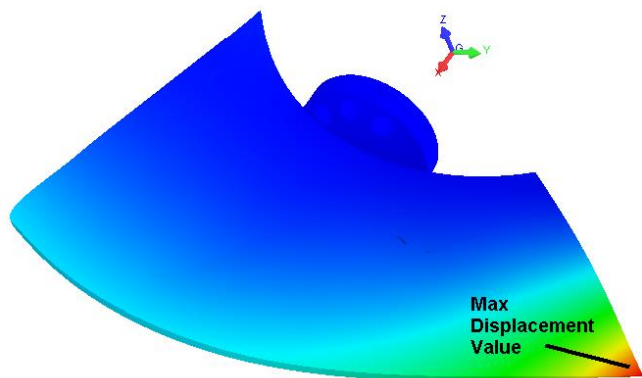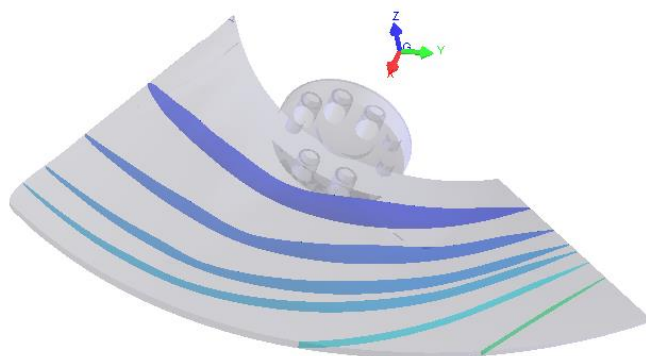

Fig. 7 The displacement distribution

Fig. 8 The displacement distribution with iso values

8. Conclusions

The mixed software Autodesk Inventor and Cosmos Design Star offer to the designer engineer a numerical solution to verify the resistance of the axial turbine blade. The complex geometry of the blade must be modelled with high precision, to assure the curves continuity and correct results. The mesh in the sensible zones must be created much smaller for a good accuracy of the final results.

References

1. **Mănescu, T., Nedelcu, D.** - *Analiză structurală prin metoda elementului fînit*, Editura "Orizonturi Universitare", Timişoara 2005.
2. **Nedelcu, D.** - *Modelare parametrică prin Autodesk Inventor*, Editura "Orizonturi Universitare", Timişoara 2004.
3. **Nedelcu, D., Pădurean, I.** - *CAD modelling of an axial blade turbine using Autodesk Inventor*, Scientific Bulletin of the „Politehnica" University of Timisoara, Transactions on Mechanics, Tom 52(66), 2007.
4. **Structural Research & Analysis Corporation** – C*osmos Design Star. Basic User's guide*, 2003.

STAREA DE TENSIUNI ŞI DEFORMAŢII PE O PALETĂ DE TURBINĂ AXIALĂ PRIN METODA ELEMENTELOR FINITE

Rezumat

Geometria unei palete de turbine axiale este deosebit de complexă, cu grosime variabilă şi unghi variabil de la butuc către periferie. Uzual, valorile maxime ale tensiunii sunt localizate în zona butucului. Metodele analitice oferă soluţii aproximative. Metodele experimentale pot fi utilizate, dar cu costuri ridicate legate de construcţiei paletei model şi a aparaturii de măsură. Ca o alternativă, se poate utiliza metoda numerică de analiză cu elemente fînite. Lucrarea descrie procedura de calcul a metodei, utilizată pentru a calcula starea de tensiuni şi deformaţii pentru o paletă de turbină axială, utilizând combinaţia de programe Autodesk Inventor şi Cosmos Design Star.

Referenţi ştiinţifici: Prof. dr. ing. Mircea POPOVICIU
Prof. dr. ing. Victor BĂLĂŞOIU