Application of a 2-D approximation technique for solving stress analyses problem in FEM

by

H. Khawaja

reprinted from

THE INTERNATIONAL JOURNAL OF
MULTIPHYSICS

2015: VOLUME 9 NUMBER 4

MULTI-SCIENCE PUBLISHING

Application of a 2-D approximation technique for solving stress analyses problem in FEM

H. Khawaja
UiT-The Arctic University of Norway,
Tromsø, Norway

ABSTRACT

With the advent of computational techniques and methods like finite element method, complex engineering problems are no longer difficult to solve. These methods have helped engineers and designers to simulate and solve engineering problems in much more details than possible with experimental techniques. However, applying these techniques is not a simple task and require lots of acumen, understanding, and experience in obtaining a solution that is as close to an exact solution as possible with minimum computer resources. In this work using the finite element (FE) method, stress analyzes of the low-pressure turbine of a small turbofan engine is carried out by employing two different techniques. Initially, a complete solid model of the turbine is prepared which is then finite element modelled with the eight-node brick element. Stresses are calculated using this model. Subsequently, the same turbine is modelled with four-node shell element for calculation of stresses. Material properties, applied loads (inertial, aerodynamic, and thermal), and constraints were same for both the cases. Authors have developed a “2-D approximation technique” to approximate a 3-D problem into a 2-D problem to study the saving invaluable computational time and resources. In this statistics technique, the 3-D domain of variable thickness is divided into many small areas of constant thickness. It is ensured that the value of the thickness for each sub-area is the correct representative thickness of that sub area, and it is within three sigma limit. The results revealed that technique developed is accurate, less time consuming and computational effort saving; the stresses obtained by 2-D technique are within five percent of 3-D results. The solution is obtained in CPU time which is six times less than the 3-D model. Similarly, the number of nodes and elements are more than ten times less than that of the 3-D model. ANSYS ® was used in this work.

1. INTRODUCTION

Stress analysis in any engineering field is invariably complex, and obtaining an exact solution is almost impossible if the geometry is complicated. In such a situation, engineers usually resort to numerical methods to solve the problems. With the advent of computers, one of the most powerful techniques that have been developed in the realm of engineering analysis is the FE method, and the method, being general, can be used for the analysis of structures of complex shapes and complicated boundary conditions [1, 2]. However employing this method correctly is in itself a challenging task.

---

*Corresponding Author: E-mail: hassan.a.khawaja@uit.no

Today computational power is much larger, more reliable, and relatively cheap and as most technology related setups have access to computers, the popularity of using numerical methods is an ever increasing phenomenon. Especially FE method is being used to a large extent for structural analysis for example [3–6]. Description of the development of the FE method and its application is thoroughly discussed.

In this effort, we have tried to solve the complex 3-dimensional problem of a low-pressure turbine first by using '2-dimensional approximation technique' to take advantage of reduced computational time and ease of finite element modeling and meshing. The same problem was subsequently solved modeling the blade with 3-D, 8 noded brick type of finite element. After that comparison of the result is made and discussed. For this work commercially available FE software ANSYS® was used as a tool for the analysis [7–9].

2. 2-DIMENSIONAL APPROXIMATION TECHNIQUE

2-Dimensional approximation requires a bit of modeling work. However, its finite element model is quite easy to build and easy to mesh compared to the 3-dimensional model. 3-Dimensional CAD model of turbine blade was built as per the recommendation of aero-propulsion experts. Its 3-dimensional shape is shown "Figure 1". There are 59 blades in the turbine joined with a solid hub. Blades have thicker aerofoil at the hub in comparison to tip. Also, the geometric and aerodynamic twist is there to have a best possible performance from the turbine.

The 2-dimensional approximation was applied to the turbine blades. The entire blade was distributed into many areas. In 2-D model, it was assumed that the thickness is constant in each area. The value of that thickness was, in fact, the average thickness of that area. Multiple areas were built keeping the constant value of thickness, which was the average value of the actual variable thicknesses of the blade in chord and span directions as illustrated "Figure 2", where A, B, C, D, E, and F are thicknesses associated with one area as shown. Also standard deviations of thicknesses were calculated for respective areas and tried to be maintained within limits.

The maximum value of thickness, the minimum value of thickness, the average value of thickness and standard deviation from the average value for each area within the blade is shown "Table 1". Areas built for turbine blade using 2-D approximation technique are shown "Figure 3". Turbine disc (Hub) is approximated by an area having a constant value of thickness equal to 3 mm.

Division of areas were made in such a way that maximum and a minimum value of thicknesses in an area remains within 3-sigma limits. After the determination of thicknesses,

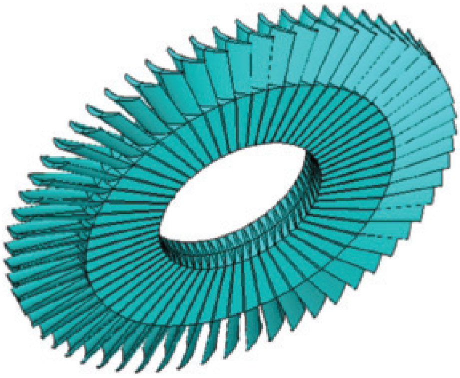


Figure 1: CAD model of turbine

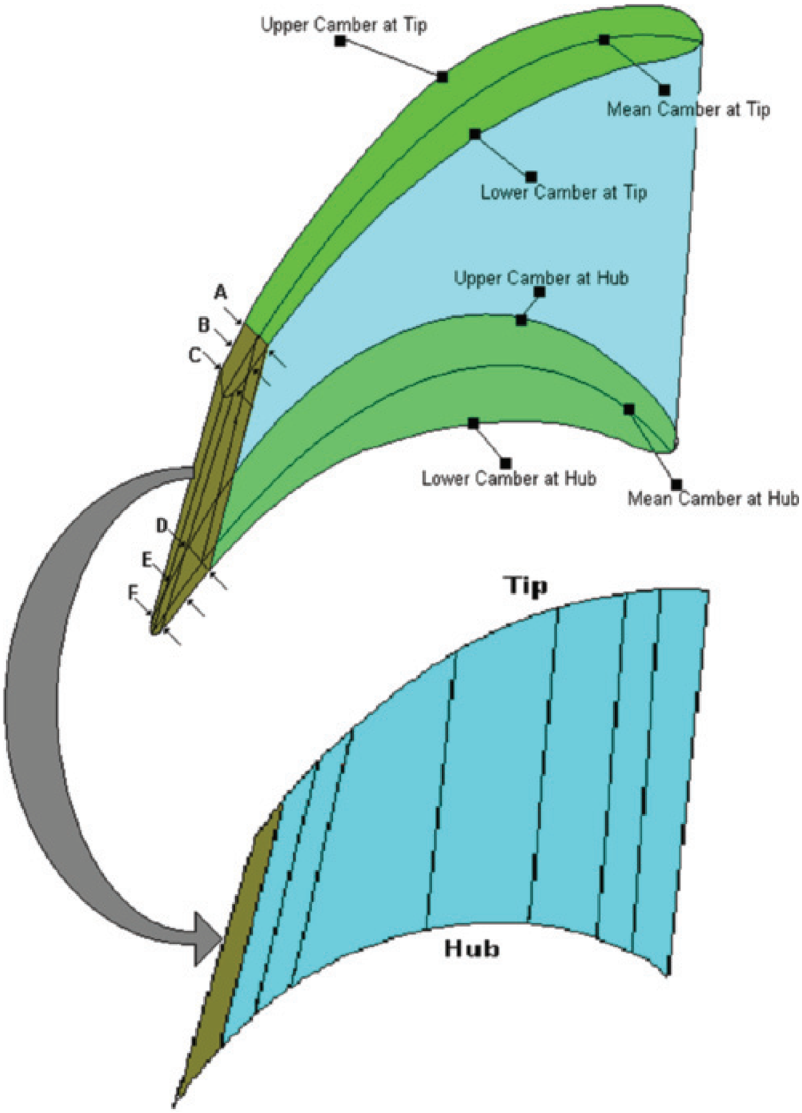


Figure 2: Illustration of thicknesses taken from 3-D turbine blade for making 2-D areas

the mean chamber line of tip and root aerofoil were modeled and joined, keeping areas of different thicknesses as separate. These areas were joined by glue command in ANSYS® modeling module. Turbine model built using 2-dimensional approximation is shown "Figure 4".

3. FINITE ELEMENT MODELING AND APPLIED CONSTRAINTS

The 2-D approximated model of the turbine was meshed with 4 Node Shell 63 element type "Figure 5a". Node coupling was done at blade-hub Joint "Figure 6". Mesh sensitivity analysis was carried out to have a mesh with an optimum number of nodes. The optimized mesh is capable of giving accurate results with minimum utilization of computational resources, which saves time.

Table 1: Thicknesses associated with 2-D approximated areas

Area No.	Maximum Thickness (mm)	Minimum Thickness (mm)	Average Thickness (mm)	Standard Deviation(mm)
1	0.845	0.811	0.828	0.012
2	1.804	1.771	1.787	0.011
3	2.130	1.928	2.029	0.071
4	2.666	2.212	2.439	0.160
5	2.982	2.293	2.638	0.243
6	2.589	1.975	2.282	0.217
7	2.075	1.674	1.875	0.142
8	0.939	0.782	0.860	0.055

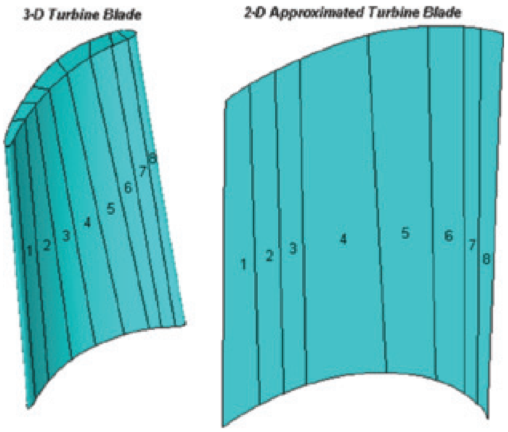


Figure 3: 3-D Turbine blade and its 2-D approximated form

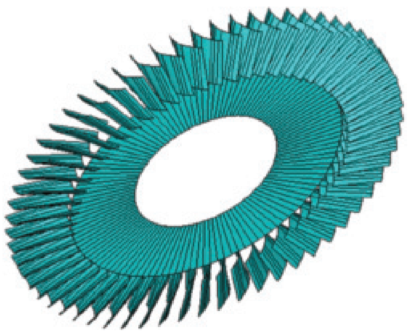
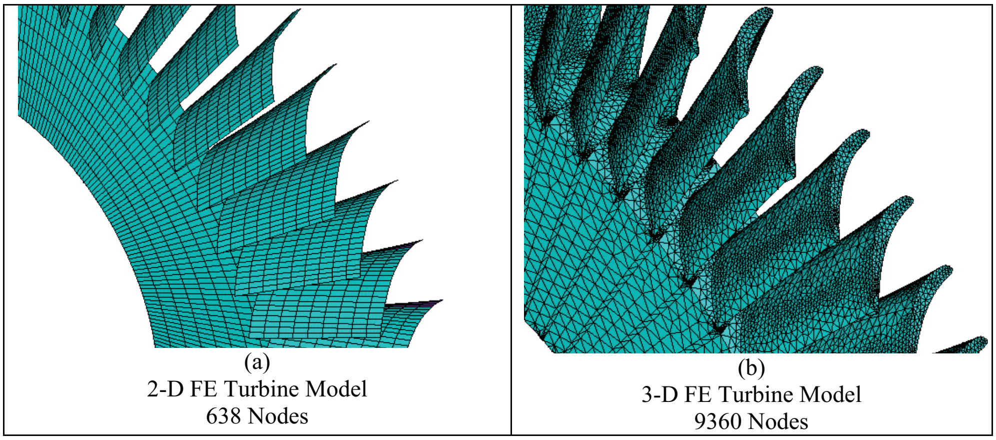


Figure 4: CAD model of 2-D approximated turbine

3-D model of the blade was divided into many small volumes, which were glued and meshed with an 8 Node Brick 45 element type “Figure 5b”. Mesh sensitivity analysis was also carried out on 3-D FE model of the turbine. The optimized mesh was selected for analysis performed ahead.


(a)
2-D FE Turbine Model
638 Nodes

(b)
3-D FE Turbine Model
9360 Nodes

Figure 5: (a) & (b) Finite element model of turbine

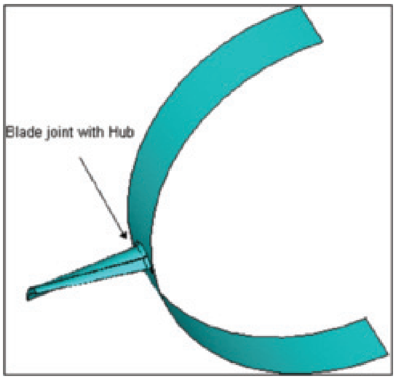


Figure 6: Illustration of blade hub joint (single blade)

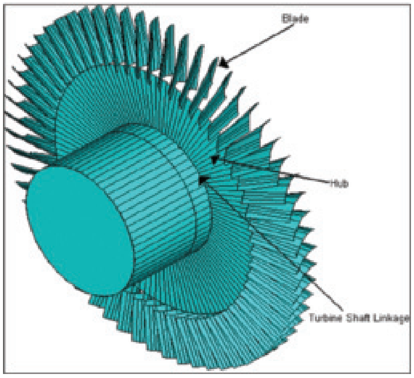


Figure 7: Illustration of turbine shaft linkage

Before running FE solution, displacement constraints are also required to be defined; for that all nodes at turbine-shaft linkage “Figure 7” were constrained in all DOFs (Degree of Freedoms). Both analyzes were carried out using linear elastic material model. Mechanical properties of Nickel based alloy Inconel 718 were taken for analysis of turbine [10].

4. LINEAR STATIC ANALYSIS AND APPLIED LOADS

The static analysis calculates the effects of steady loading conditions on a structure. A static analysis can, however, include steady inertia loads (such as gravity and rotational velocity), and time-varying loads that can be approximated as equivalent static loads.

Static analysis is used to determine the displacements, stresses, strains, and reaction forces in structure or components caused by applied loads. The kinds of loading that can be applied in static analysis include:

- Externally applied forces and pressures
- Steady state inertial forces (such as gravity or rotational velocity)
- Imposed (nonzero) displacements
- Temperature (for thermal strain)

As turbine blades are subjected to various types of loadings such as aerodynamic, thermal and inertial loads. These loads as specified "Table 2" were applied. Aerodynamic load, which is the difference of pressure acting on the upper and the lower surface of the blade, was applied as distributed load on the lower surface of the turbine blade. The thermal load was applied to the entire blade, and the inertial load was applied as body load that is due to the RPM of the turbine.

Table 2: Loading conditions on turbine as per defined cases

CASE 1	Maximum Pressure	0.5745 MPa
(Normal Operation)	Maximum Temperature	1056.55 K
	Operating RPM (Inertia)	32200 RPM
CASE 2	Maximum Pressure	0.6320 MPa
(Over-run Operation)	Maximum Temperature	1083.33 K
	Operating RPM (Inertia)	38000 RPM
CASE 3	Maximum Pressure	0.4309 MPa
	Maximum Temperature	972.22 K
	Operating RPM (Inertia)	32200 RPM
CASE 4	Maximum Pressure	0.5027 MPa
	Maximum Temperature	1027.78 K
	Operating RPM (Inertia)	38000 RPM

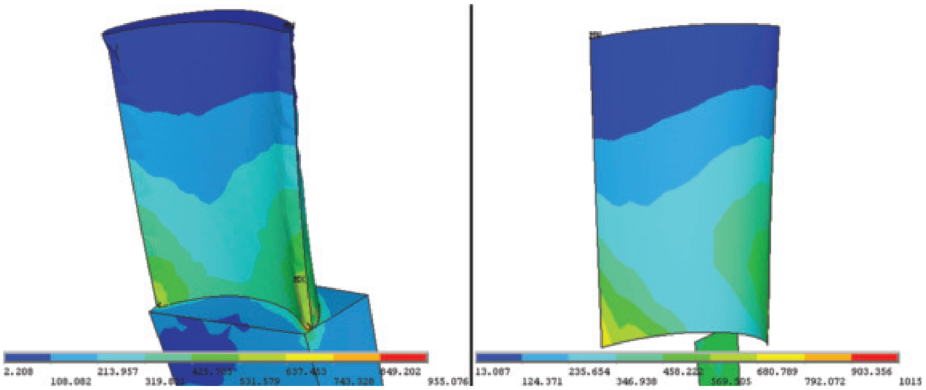


Figure 8: Effective stress (MPa) contour plot of turbine blade for normal operation

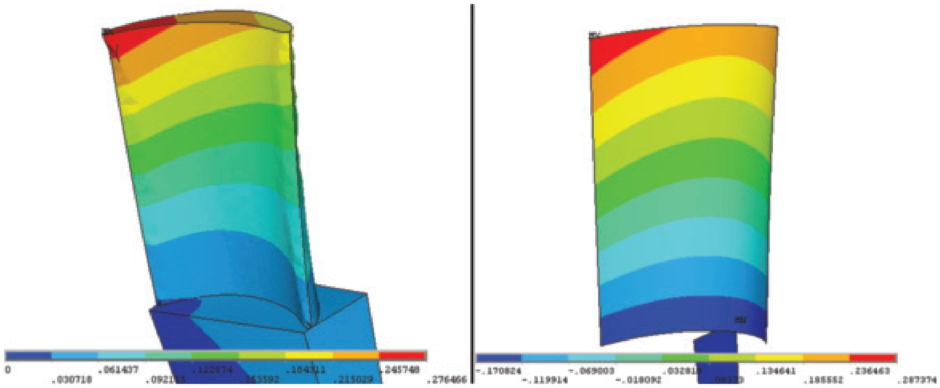


Figure 9: Displacement (mm) contour plot of turbine blade for normal operation

Table 3: Comparison of results obtained by stress analysis

Load Cases	Max. Values	3-D blade	2-D blade	% Difference
CASE 1	Stress (MPa)	955	1015	6.30 %
(Normal Operation)	Displacement (mm)	0.276	0.287	4.00 %
CASE 2	Stress (MPa)	1077	1127	4.64 %
(Over-run Operation)	Displacement (mm)	0.314	0.336	7.01 %
CASE 3	Stress (MPa)	716	761	6.30 %
	Displacement (mm)	0.211	0.215	1.90 %
CASE 4	Stress (MPa)	846	898	6.15 %
	Displacement (mm)	0.242	0.251	3.72 %

5. LINEAR STATIC ANALYSIS RESULTS

The nodes required by the 2-dimensional approximate model are almost 15 times lesser than the number of nodes required for analyzing the 3-dimensional model, and it was found out that solution time is six times lesser for 2-dimensional approximated model than the 3-D model. Von-Mises (Effective) stress contour plots of 2-dimensional approximated and 3-dimensional blade as per static analysis for normal operational load are shown “Figure 8”. Also, displacement contour plots of both the blades under normal operational load are given “Figure 9”. Comparison of results is given “Table 3”. The difference between 2-D model and 3-D model for normal operation for maximum stress is 6.30 % where as far as maximum displacements it is 4.00 %.

6. CONCLUSION

Although, accuracy in results even by half a percent is appreciated but sometimes the purpose of the analysis is just to build an idea about the results. Under such a situation, the analyzer strives to have results in an easiest and fastest possible way. ‘2-dimensional approximation’ technique is one of such methods that will help the analyzer to solve the problem in the least possible time with some errors as discussed.

REFERENCES

1. Cook, R. D., *Concepts and applications of finite element analysis*. 2001: Wiley.
2. Ross, C. T. F., *Finite Element Methods in Structural Mechanics*. 2001: Ellis Horwood Limited.

3. Khawaja, H. and Moatamedi, M. *Selection of High Performance Alloy for Gas Turbine Blade Using Multiphysics Analysis*. The International Journal of Multiphysics, 2014. 8(1): pp. 91–100.
4. Khawaja, H. and Parvez, K. *Validation of normal and frictional contact models of spherical bodies by FEM analysis*. The International Journal of Multiphysics, 2010. 4(2): pp. 175–185.
5. Khawaja, H., et al., *Optimization of elstomeric micro-fluidic valve dimensions using non-linear finite element methods*. The International Journal of Multiphysics, 2009. 3(2): pp. 187–200.
6. Khawaja, H. A., et al., *Study of CRFP Shell Structures under Dynamic Loading in Shock Tube Setup*. Journal of Structures, 2014. 2014: p. 6.
7. ANSYS®, *Academic Research*. release 14.0.
8. ANSYS®, *Academic Research, Theory Reference*, in *Mechanical APDL Guide*. release 14.0.
9. Stolarski, T., Nakasone, Y. and Yoshimoto, S. *Engineering Analysis with ANSYS Software*. 2011: Elsevier Science.
10. *Material Handbook of High Performance Alloys*. 2014, Special Metals Corporation: http://www.specialmetals.com.