Analysis of centrifugal stresses in the rotor of a cryogenic turboexpander

Parthasarathi Ghosh, Sunil Kr Sarangi

Cryogenic Engineering Centre, IIT Kharagpur 721 302, India

Cryogenic turbines, because of moderate to high-pressure ratio and low flow rate, rotate at high speeds with peripheral speeds approaching the velocity of sound at prevailing temperatures. At such high rotational speeds, significant centrifugal stresses are generated in the large diameter components – turbine wheel, compressor impeller and shaft collar. The blades of the wheels, which are thin and curved, also expand and deflect, reducing clearances and distorting flow passages. This paper presents an FEM analysis of the stress field and the deflection pattern in the rotor of a high-speed cryogenic turbine, and a recommendation for an appropriate design strategy.

INTRODUCTION

Because the strength of most materials improves at low temperature, the general perception among engineers and scientists is that stress considerations are unimportant in case of cryogenic turbines. On the other hand, cryogenic turbines, because of the moderate-to-high pressure ratio and low flow rates, rotate at high speeds leading to significant centrifugal stresses in the rotor.

A finite element analysis of a complete cryogenic turbine rotor has been carried out using the ALGOR FEM package. The analysis provides a map of the stress field and the deflection pattern in the critical components under design conditions. The effects of geometric features such as blade thickness and fillet radii on the maximum stress have been brought out. The paper presents the formulation of the problem, boundary conditions, analysis and results. A clear recommendation is made on the choice of materials and dimensions of relevant structural features to keep the stresses and deflections within acceptable limits.

DESCRIPTION OF THE ROTOR

The specifications of the turbine system and the dimensions of the components have been obtained from the design of a prototype turboexpander developed at IIT, Kharagpur [1].The schematic of the rotor is shown in the Fig 1. It consists of the shaft, the turbine wheel and the brake compressor impeller. The compressor is used to load the turbine, which produces power by expanding high pressure gas to a lower pressure. The wheel and the impeller are attached to the shaft at both ends "socket head button" screws. The expected rotational speed of the rotor at the designed condition is 140,000 r/min. The shaft is supported by two aerostatic journal bearings and two aerostatic thrust bearings. Orientation of the rotor is vertical with the brake impeller located at the upper side of the assembly.

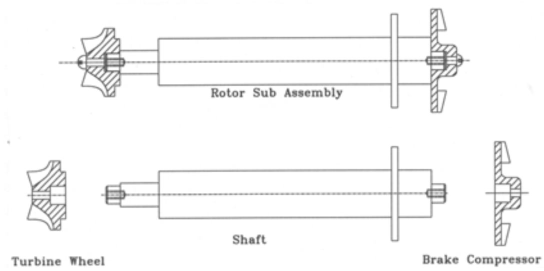

Figure 1: Schematic of the rotor to be analysed

OBJECTIVES OF THE ANALYSIS

A static stress analysis has been performed on the turbine rotor with the following objectives:

1. to map the stress field in the rotating components and to identify points of high stress for possible modification of design,
2. to obtain the deflection pattern under centrifugal load at the nominal rotational speed (The displacement pattern reveals the radial growth of the rotor under centrifugal stress. The deflected shape affects the clearances between the rotating components and the corresponding static components), and
3. to analyse the effect of geometric features such as thickness distribution of the turbine blades and radius of the fillets on the shaft collar on the computed stress pattern.

SOLUTION STRATEGY

The stress analysis has been carried out using a solid model of the rotor, followed by finite element meshing, application of loads and boundary conditions and finally by solution of the resulting finite element equations. The FEM results have been compared with those obtained from simple formulas given in standard textbooks. The detailed steps in the FEM analysis process are :

1. Creation of a solid model using a parametric solid modeling software. [We have used AutoCAD Mechanical Desktop Release 3 and 4 for this step.]
2. Meshing of the solid model (*.dwg file) so generated using the meshing function of the ALGOR finite element analysis package. [An extender provided by ALGOR has established a seamless relation between the AutoCAD MDT4 solid modeller and the ALGOR FEM software. The coordinates of the elements, generated after meshing, are saved in a *.ami file which is exported to the main ALGOR FEM software.]
3. Solution of the finite element equations to yield the complete stress and deflection field.

STRESS ANALYSIS OF THE TURBINE WHEEL

Figure 2 describes the geometry of the turbine wheel used for analysis. The other features of the turbine geometry are:

1. Radial blading;
2. A constant tangential thickness of 1 mm;
3. No taper along the hub-to-shroud direction;
4. Constant hub fillet radius of 0.5 mm.
5. Material of the wheel is Al 6061 T6.

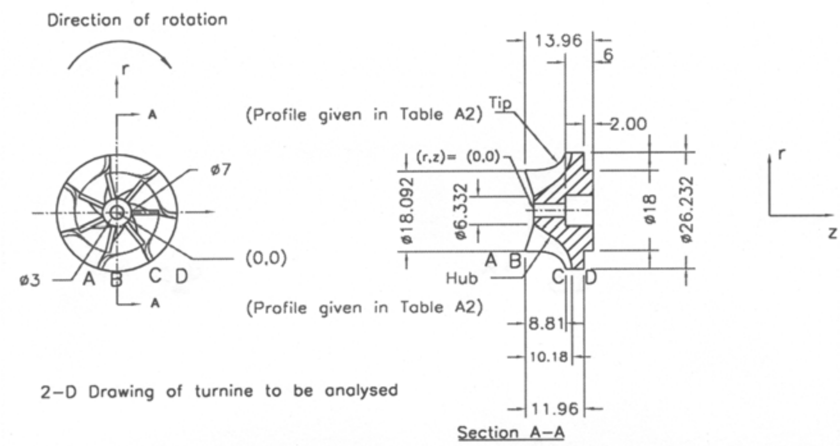

Figure 2 Geometry of the turbine wheel used in the analysis

Geometric modelling of the wheel

Due to an inherent mismatch between the MDT and ALGOR, we followed a somewhat complex route to create the solid model in MDT. A wire frame was created in MDT 3.0 and transferred to ALGOR Superdraw-III using IGES translation. A closed surface was created using the ALGOR surface modeller Supersurf, and was transported to Superdraw-III. The surface mesh was further enhanced using the Merlin Meshing Technology (MMT), a feature provided by the ALGOR package. The turbine wheel has a 3 mm diameter hole with overall external diameter of 26.3 mm. Discontinuities exist at the bore and at the blade-hub intersections. To take into account these features, the surface mesh was refined using the “refinement near short” and ‘refinement near gaps” commands of MMT. Quadrilateral mesh type was chosen.

The surface mesh was then transformed into solid mesh using the routine HEXAGEN. For the solid mesh, the maximum aspect ratio has been chosen as 100 and the wrap angle limit has been set at 30^0 (maximum). Figure 3 shows the meshed geometry (pie-slice model) of the wheel, projected on the x-y plane, as seen from the reverse side. The origin is at O. The lines AB and CD are the cut surfaces projected on the x-y plane, making angles of 150° and 70° with the x-axis respectively.

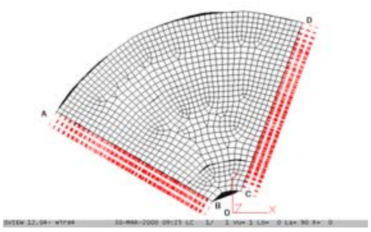

Figure 3 The meshed geometry of the turbine wheel

Displacement boundary conditions

Elastic translation boundary conditions have been chosen on symmetry surfaces. The stiffness vectors on the cut surface AB is inclined to the X axis by 240^O, and are give as

$$\hat{n} = -0.5\,\hat{i} - 0.866\,\hat{j}$$

while that on the surface CD are given as (inclination to x axis = -20^O)

$$\hat{n} = 0.84\,\hat{i} - 0.529\,\hat{j}$$

Application of loads

Centrifugal load (mϖ^2r) corresponding to a rotational speed of 1,40,000 r/min about the z-axis is applied to all elements. We have neglected the bending load generated due to the pressure difference at the suction and pressure surfaces.

The turbine was modeled using 3-D solid non-conformal brick elements. Typical solution time for the analysis was three hours on a P-II based PC and the process consumes 34 MB of disk space.

Analysis of the results

Figure 4 shows the displacement profile of the model. Table 1 gives the displacements of a few important points on the wheel surface. These points have been identified in Fig. 3. From the displacement plots the following may be observed:

1. As expected, there is no tangential displacement on the surfaces of symmetry (cut surfaces). At the outer diameter the displacement is fully radial.
2. Displacement is the highest at the exit tip of the blade. There is a radial growth of the wheel, which decreases the clearance between the wheel and the shroud. The radial displacement is 14 μm at 1,40,000 r/min. The clearance between the turbine and the shroud has been kept as 200 μm as a first approximation. From the vibration study of the machine, the maximum vibration amplitude has been found to be about 5 μm at the rotational speed of 100,000 r/min [1]. The geometric run-out of the rotor has been measured to be 10μm. An increase in the clearance is expected at low temperature due to differential shrinkage of aluminium wheel and the stainless steel shroud. Considering the above observations, it can be concluded that the shroud clearance at the warm conditions can be reduced to 100 μm for a gain in efficiency at the design conditions.
3. It can be seen, from Fig. 4 and Table 1 that there is considerable tangential displacement at the tip of the blade. This is due to straightening of the blade due to centrifugal forces. Thus, we find that the fluid passage between the blades changes under running condition and this may be taken into consideration in design stage.

Figure 5 shows the stress field on the turbine wheel. The critical regions from a stress point of view are the hub-to-blade intersection and the bore region. In addition, the following conclusions may made from the stress plots.

1. The maximum stress as seen from the back plate occurs at the rim of the counter bore, and has a value of 74 MPa. If the turbine base plate is considered as a uniform disk of diameter 13.16 mm, with a central hole of diameter 3.5 mm, the maximum stress is predicted by formula

$$\sigma_{\theta\max} = \left(\frac{\rho\omega^2}{4}\right)\left[(1-\nu)R_1^{\,2} + (3+\nu)R_2^{\,2}\right] = 84.0\ MPa$$

   The difference between the two results can be traced to the assumption of plane stress conditions in the latter analysis.
2. In the bladed region, the maximum stress occurs on the blade-hub interface over the convex surface of the blade. High values of stress can be observed up to a radius of $0.4D_2$. The highest stress, however, remains well below the yield stress of the material (270 MPa).

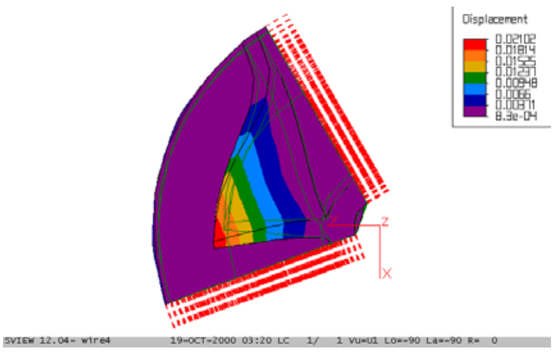

Figure 4 Displacement plot of turbine wheel

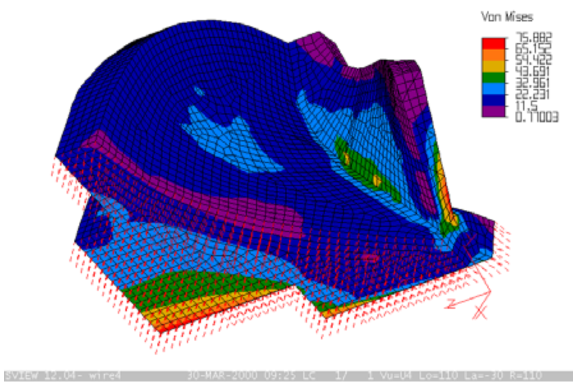

Figure 5 Stress plot of turbine wheel

STRESS ANALYSIS OF SHAFT AND BRAKE COMPRESSOR

A major part of the stress analysis of the shaft and the brake compressor has been reported in a previous paper [2]. Some of the major findings which have not been reported are:

1. The highest stress in the shaft occurs at the shaft-collar intersection. The maximum stress is 225.5 MPa with a fillet of radius 0.2 mm. We have changed the fillet radius to 1.0 mm and analysed the stress pattern. The maximum stress has been reduced to 197 MPa (material yield stress 215 MPa), which makes the design reasonably safe.

Table 1 Displacement at important points on the turbine wheel.

Point	Coordinates					Displacements				
	Cartesian			Polar		Cartesian			Polar	
	x(mm)	y(mm)	z(mm)	r(mm)	θ (deg)	dx(mm)	dy(mm)	dz(mm)	dr(mm)	θ (deg)
A	0.23	9.18	-1.21	9.19	88.56	0.011	0.005	0.017	0.012	-0.068
B	-0.23	8.91	-2.06	8.91	-88.51	0.012	0.009	0.015	0.015	-0.079
C	-11.05	7.07	7.18	13.13	-32.61	-0.002	0.002	-0.001	0.003	-0.003
D	-10.49	7.90	7.18	13.13	-36.98	-0.002	0.002	-0.001	0.003	-0.002

2. As the possibility of yielding of the material cannot be confidently ruled out, it is recommended that a stronger material e.g. K-Monel be used for the shaft instead of SS-304. Attempt should also be made to reduce the diameter of the collar and reducing the mass of the collar by cutting a circumferential slit while providing adequate bearing area to the thrust bearings. A more liberal fillet radius is also expected to make a difference.
3. The blade to hub section of the brake compressor requires appropriate fillet distribution to avoid high stress.

REFERENCES

1. Ghosh, P., Analytical and Experimental Studies on a Cryogenic Expansion Turbine, Ph. D Thesis IIT Kharagpur (2002)
2. Ghosh, P. and Sarangi, S., Static Stress Analysis of a Cryogenic Turboexpander Using Finite Element Method International Cryogenic Engineering Conference-18 Narosa Publishers, Mumbai, India (2000).