ANALELE UNIVERSITĂȚII
"EFTIMIE MURGU" REȘIȚA
ANUL XX, NR. 1, 2013, ISSN 1453 - 7397

Ana-Maria Budai

Study of Finite Element Number Influence over the Obtained Results in Finite Element Analyses of a Mechanical Structure

This paper present the results of a study that was made to establish the influence of finite element number used to determined the real load of a structure. Actually, the study represent a linear static analyze for a link gear control mechanism of a Kaplan turbine. The all analyze was made for the normal condition of functioning having like final scope to determine de life time duration of mentioned mechanism.

Keywords*: linear, static, analyze, link, finite element*

1. Introduction

Higher and higher structures complexity, their exploitation condition much stressing lately and constant necessity of exploitation conditions optimization made the classical strength calculus not satisfied.

Finite element method, at that moment, represent the most used numerical method to calculated any structure in any stress conditions. The linear static analyzes using finite element method presume the establish of two aspects:

1) the digitization mode for calculus model (the type, form and number of digitization elements and also the digitization net nodes number);
2) the analyze approach method.

Through informational point of view the finite element model used must permit the most precise information's volume modeling. For the most precise results it is necessary that, function of geometrical model used for digitization, to have a maximum possible number of nodes (Figure 1). The digitization model must also complain two others conditions: every nodes must have the maximum possible degree of freedom and the interpolation function to have the biggest number of parameters possible.

Frequently, to realize such kind of analyze, can be used one of the next two methods:

- h-element method;
- p-element method.

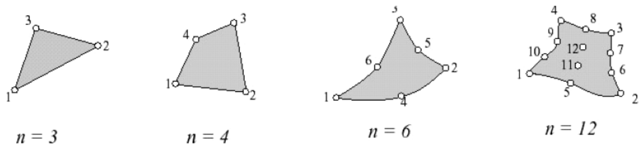


Figure 1. Geometrical variety of digitization elements and the possible nodes attach to them

The classical method used for digitization is the h-method. To increase the results accuracy, when the h-method is used, must use a digitization net with the highest density. For the p-method, to provide highest results accuracy, is necessary to increase the multinomial grade. The analyze was made using the classical h-method.

2. Analysis

Linear static analyze for gear control link mechanism was made in collaboration with a specialists group from CCHAPT. In Figure 2 are shown the assembly loading (a.) and the restriction applied to the geometrical model of link (b). Significance of notation used in below figure are:

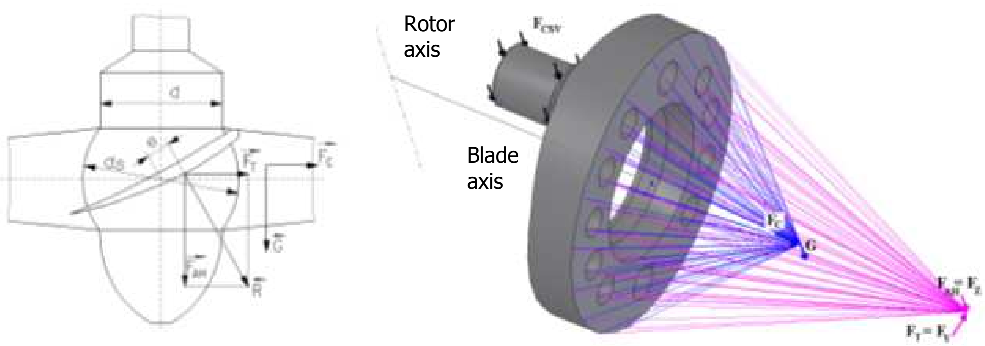
- the gravity force of assembly blade-mandrel-link G = 286900 N;
- the centrifugal force F_c corresponding to a nominal speed by 71.43 rot/min;
- the axial force on the blade F_{AH} resulted from tests on model;
- the tangential force on blade F_T.

The strength calculus was made for the following loadings:

- two variants of power actuator: the force F_{cvs}=4641221 N respectively F_{cvs}=3100000 N which was resulted from cinematic test;
- a centrifugal force corresponding to the nominal speed 71.43 rot/min by F_c=3992315 N applied in assembly center of gravity. The center of gravity have the following coordinates: x=2487.2 mm, y=33 mm, z=-80.6 mm.

The other values used to realize the analyze are shown in Table1. The numerical results obtained are centralized in Table 2. The entire analysis was made using the finite element analyze software COSMOS DESIGN STAR.

To obtained viable conclusions the study was made for four calculus variants with four different density of mesh as is shown in Figure 3.

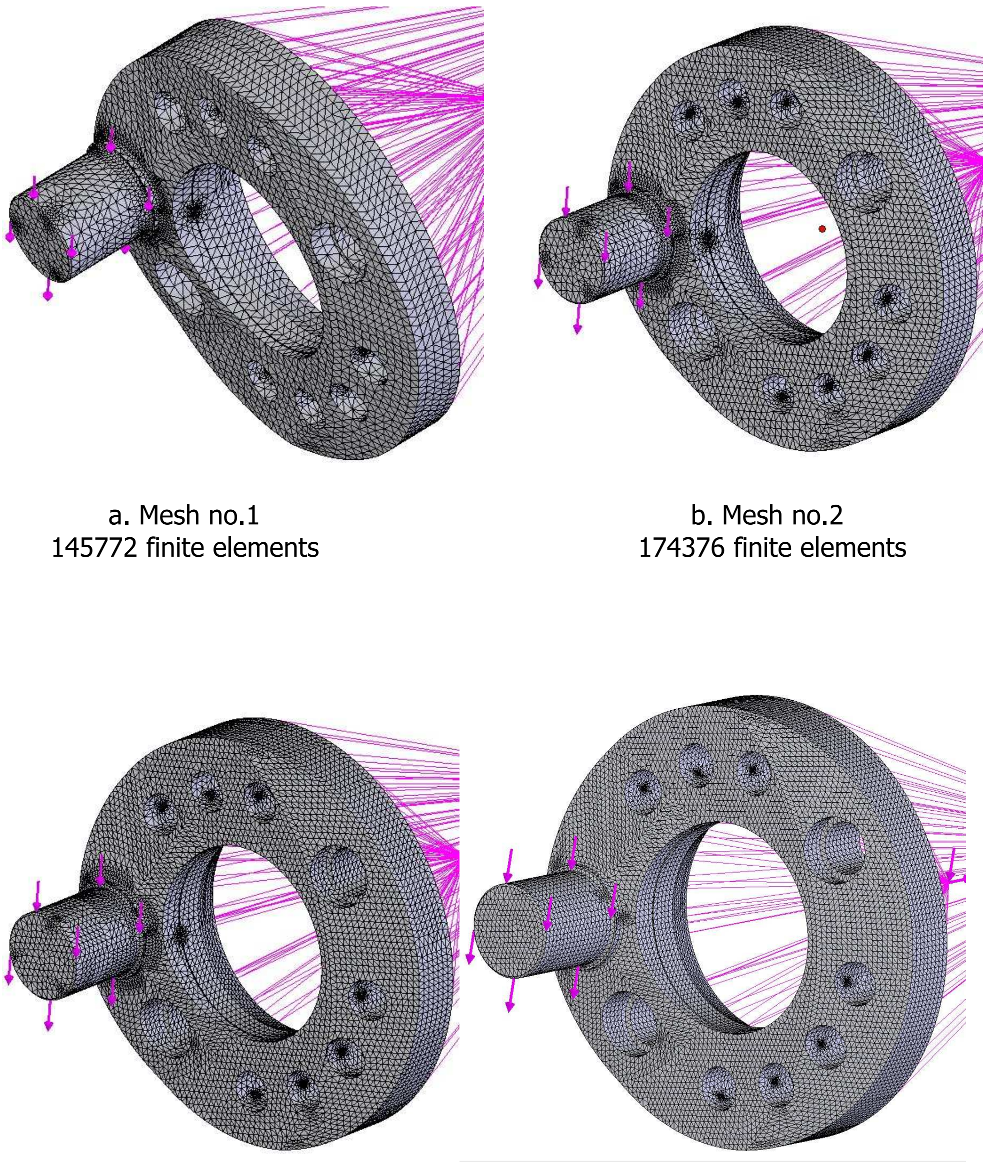


a. b.

Figure 2. The loading diagram

Table 1

The link loaded					
Centrifugal force [N]					3992315
Gravity force for the assambly [N]					286900
Case	**Blade angle [º]**	**H [m]**	**Axial force F_{ax}[N]**	**Tangential force F_t[N]**	**Maximum power actuator $F_{sv,max}$ [N]**
1	+17.5	25	1775041	1336898	4641221
1	+17.5	25	1775041	1336898	3100000
2	+10	25	1771437	1024955	4641222
2	+10	25	1771437	1024955	3100000
3	+10	31.4	1887115	1292381	4641223
3	+10	31.4	1887115	1292381	3100000



a. Mesh no.1
145772 finite elements

b. Mesh no.2
174376 finite elements

c. Mesh no.3
258779 finite elements

d. Mesh no.4
400750 finite elements

Figure 3. The meshing variants used for analysis

Table 2

Power actuator F_{csv} [N]	Var	Global Mesh Size (GMS)	Finite elements number	φ=+17.5° Case 1 F_{ax}= 1775041 N F_T= 1336898 N		φ=+10° Case 2 F_{ax}= 1771437 N F_T= 1024955 N		φ=+10° Case 3 F_{ax}= 1887115 N F_T= 1292381 N	
				Sigma VonMises max [MPa]	Maximum elongation [mm]	Sigma VonMises max [MPa]	Maximum elongation [mm]	Sigma VonMises max [MPa]	Maximum elongation [mm]
4641221	1	36	145772	563.8	1.118	540.4	1.109	541	1.075
	2	30	174376	578.2	1.098	559	1.091	561.6	1.069
	3	25	258779	584.2	1.127	560.2	1.116	559.6	1.095
	4	20	400750	585.6	1.175	560.9	1.162	557.8	1.136
3100000	1	36	145772	395.5	0.7741	382.7	0.765	387.6	0.7417
	2	30	174376	413.02	0.7688	401.8	0.7701	407	0.7607
	3	25	258779	413.7	0.7841	398.6	0.7836	403.1	0.7732
	4	20	400750	411.5	0.8143	394.3	0.8111	394.1	0.8

3. Conclusion

After experimental test two important conclusions were revealed:

- along with increasing of meshing net density the accuracy of results it is also grows by registering an increase of estimate strength between 10 and 25 MPa;
- can be observe that as the power actuator increasing the difference between stress value to minimal density of mesh and the one to the maximum density grows.

Finally, can be concluded that to the mechanical systems intense demand in operation it is recommended that to finite element analyze to be used, for meshing, mesh networks with the maximum possible density.

In this way, designer engineer can made a safety designing, and the obtained structure will respond surely to the requirements imposed. To design at the limit of safety limit provide material economy but involve risks related to life time duration of structures and safety in operation.

References

[1] Câmpian V.C., Nedelcu D., Pittner (Budai) A.M., Isbăşoiu C., Alămoreanu E., *Calculul solicitărilor mecanice compuse care apar pe manivela rotorului turbinei de la CHE Porţile de Fier I. Analiza statică liniară*, Contract de cercetare, Raport Tehnic nr. U-08-400-262, 2008.
[2] Câmpian C.V., Nedelcu D., Pittner (Budai) A.M., Cojocaru V., Cuzmoş A., Gruionu L., *Calcule de durată de viaţă pentru manivela paletei rotorului turbinei de la CHE Porţile de Fier I,* Contract de cercetare, Raport U-09-400-289, 2009.
[3] Câmpian V.C., Frunzăverde D., Gruionu L., Pittner (Budai) A.M., Cuzmoş A., *Calcule de rezistenţă şi oboseală pentru manivela şi bucşele mecanismului de reglare ale paletelor rotorice ale turbinelor de la CHE Porţile de Fier I efectuate cu programele Solid Works Motion şi Solid Work Simulation*, Contract de cercetare, Raport Tehnic nr. U-10-400-302, 2010.
[4] Pacoste C. ş.a., „*Metode moderne în mecanica structurilor*", Editura Ştiinţifică şi Enciclopedică, Bucureşti, 1988.
[5] Mănescu T., Nedelcu D., *„Analiza structurală prin metoda elementului finit"*, Editura Orizonturi Universitare, Timişoara, 2005.

Address:

- Lect. Dr. Eng. Ana-Maria Budai, "Eftimie Murgu" University of Reşiţa, Piaţa Traian Vuia, nr. 1-4, 320085, Reşiţa, am.pittner@uem.ro