Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 27-28, 2011

Finite Element Analysis of Circular Plate using SolidWorks

Yeo Jin Kang* and Myung Jo Jhung
Korea Institute of Nuclear Safety, 19 Guseong-dong, Yuseong-gu, Daejeon
**s145krj@kins.re.kr*

1. Introduction

Circular plates are used extensively in mechanical engineering for nuclear reactor internal components. The examples in the reactor vessel internals are upper guide structure support plate, fuel alignment plate, lower support plate etc. To verify the structural integrity of these plates, the finite element analyses are performed, which require the development of the finite element model. Sometimes it is very costly and time-consuming to make the model especially for the beginners who start their engineering job for the structural analysis, necessitating a simple method to develop the finite element model for the pursuing structural analysis[1].

Therefore in this study, the input decks are generated for the finite element analysis of a circular plate as shown in Fig. 1, which can be used for the structural analysis such as modal analysis, response spectrum analysis, stress analysis, etc using the commercial program SolidWorks[2]. The example problems are solved and the results are included for analysts to perform easily the finite element analysis of the mechanical plate components due to various loadings.

The various results presented in this study would be helpful not only for the benchmark calculations and results comparisons but also as a part of the knowledge management for the future generation of young designers, scientists and computer analysts.

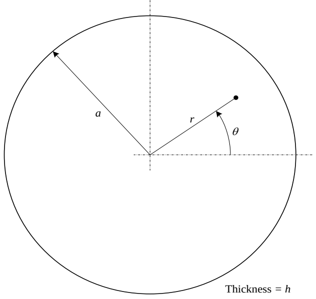

Fig. 1 Circular plate with radius *a* and thickness *h*

2. Analysis

SolidWorks is a design analysis tool based on a numerical technique called finite element analysis. SolidWorks belongs to the family of engineering analysis software products developed by SolidWorks Corp.

Consider a circular plate with a radius and a thickness of 692 mm and 3 mm, respectively. The physical properties of the material are as follows: Young's modulus = 173 GPa, Poisson's ratio = 0.3, and mass density = 8027 kg/m^3. The finite element model generated is shown in Fig. 2.

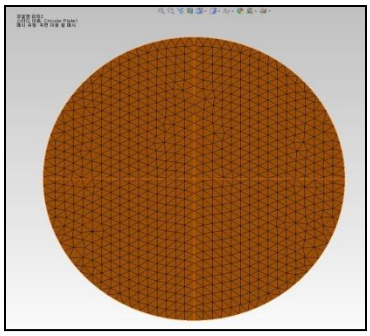

Fig. 2. Finite element model of plate

The pressure of 2.75 Pa is applied to all surface of the plate and the responses of equivalent stress and deflection are obtained as shown in Figs. 3 and 4, respectively.

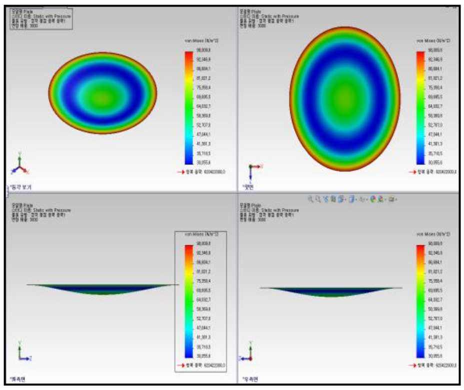

Fig. 3. Equivalent stresses due to pressure load

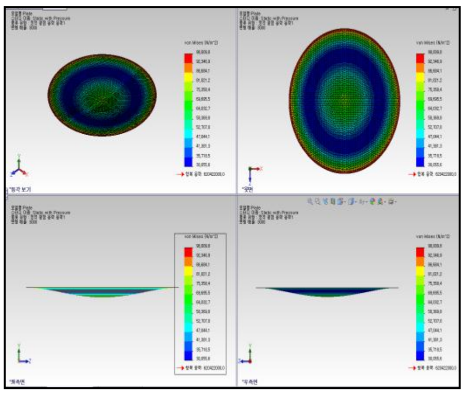

Fig. 4. Deflections due to pressure load

The unit force of 1 N is applied to the plate center and the responses of equivalent stresses are obtained as shown in Fig. 5.

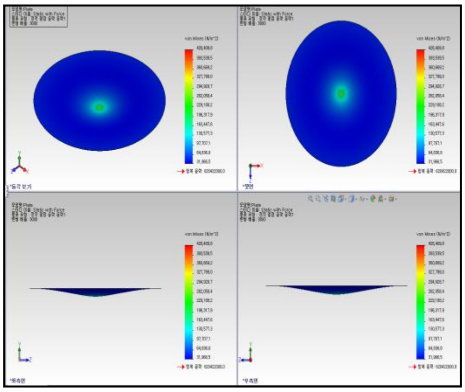

Fig. 5. Equivalent stresses due to concentrated load

The first 12 natural frequencies and mode shapes are obtained. Frequency summaries for the first 12 modes are shown in Table 1. The symbol *m* in the table represents the number of nodal circles of the mode and *n* the number of nodal diameter.

Table 1 Frequency summaries

Mode No.	Freq. (Hz)	(*m*,*n*)	Mode No.	Freq. (Hz)	(*m*,*n*)
1	14	1,0	7	71	1,3
2	30	1,1	8	71	1,3
3	30	1,1	9	85	2,1
4	49	1,2	10	85	2,1
5	49	1,2	11	98	1,4
6	56	2,0	12	98	1,4

The response spectrum of Fig. 6 is applied to the plate boundaries which are fixed in all six degrees of freedom. The responses for 2% damping ratio are combined by Square Root Sum of Squares method and equivalent stress contours are shown in Fig. 7.

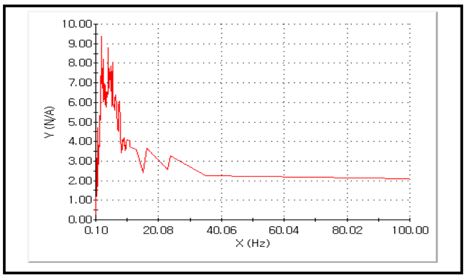

Fig. 6. Response spectrum applied

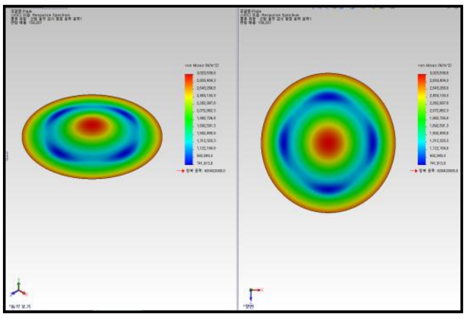

Fig. 7. Equivalent stresses by response spectrum analysis

3. Results and Discussion

Maximum deflections calculated by SolidWorks and ANSYS are compared as shown in Table 2 and there is a good agreement between them. Also, the frequencies are compared as shown in Table 3 and there is a good agreement between them, verifying the applicability of the analysis procedure and methed shown in this study.

Table 2 Deflection comparisons

Load	Max deflections (mm)	
	SolidWorks	ANSYS[3]
Pressure load	0.023	0.023
Concentrated load	0.0223	0.0222
Response spectrum	0.619	0.587

Table 3 Frequency comparisons

Mode No.	Freq. (Hz)		(*m*,*n*)
	SolidWorks	ANSYS[3]	
1	14	14	1,0
2	30	30	1,1
3	30	30	1,1
4	49	49	1,2
5	49	49	1,2
6	56	57	2,0
7	71	73	1,3
8	71	73	1,3
9	85	87	2,1
10	85	87	2,1
11	98	99	1,4
12	98	100	1,4

4. Conclusions

Using the commercial program SolidWorks in this study, static analyses due to pressure load and concentrated load are performed and response characteristics such as equivalent stress and deflection are investigated. Also, dynamic analyses such as modal analysis and response spectrum analysis are performed and their response characteristics are investigated.

The example problems are solved and the results are included for analysts to perform the finite element analysis of the mechanical plate components due to various loadings as a part of audit calculation. Deflections and frequencies between SolidWorks and ANSYS are found to be in good agreement.

REFERENCES

[1] Jhung, M.J., Choi,Y.H., Ryu,Y.H., 2009, "Free vibration analysis of circular plate with eccentric hole submerged in fluid," Nuclear Engineering and Technology, Vol. 41, No.3, pp.355-364.
[2] SolidWorks, Inc., 2011, Theory Reference for SolidWorks and SolidWorks Simulation Release 12.0.
[3] Jhung, M.J., 2010, Development of Program Plate modeler for Finite Element Analysis of Circular Plate, KINS/RR-770, Korea Institute of Nuclear Safety.