Pre-polymer 생산에 적합한 중합로 설계 및 동적해석
Analysis of Polymerization Reacting Unit by Using ANSYS/Workbench

*엄가정[1], 김상오[1] ,강대민[2], #곽재섭[2]
*Yan Jiating[1], S. O. Kim[1], D. M. Kang[2], #J. S. Kawk(jskwak5@pknu.ac.kr)[2]
[1] 부경대학교 대학원, [2]부경대학교 기계공학부

Key words : Polymerization reacting unit , ANSYS/Workbench, Vibration analysis

1. Introduction

Polyurethane products, which are often called "urethanes polymer", can be found easily in our daily lives. Especially these days, pre-polymer, which is a polymer of relatively low molecular weight that maybe mixed with compounding additives, is widely used.[1] The synthesis and characterization of NCO-terminated urethane pre-polymer, is applied more and more as a base material for the medical science, such as medical gyps mold. With the increasing use of pre-polymer, a desirable polymerization reacting unit is needed for the manufacturing of pre-polymer. In this study, simulation using ANSYS/Workbench software is applied for evaluating the polymerization reacting unit. ANSYS/Workbench is an environmental platform for handling and simulating all data involved in the analysis process. By using ANSYS/Workbench, structural and vibration analysis of the polymerization reacting unit are obtained for designing the desirable geometric shape of polymerization reacting unit later.

2. Theory Background[2]

2.1 Structural static Analysis

A static analysis in ANSYS/Workbench is used to determine the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. The overall equilibrium equation for linear structural static analysis is

$$[K]\{u\}=\{F\} \tag{1}$$

where $[K]$=Total stiffness matrix
$\{u\}$=Nodal displacement vector
$\{F\}$=Applied and reaction load vectors.

2.2 Equivalent Stress

For linear structure in ANSYS/Workbench, the stress vector is $[\sigma_x,\sigma_y,\sigma_z,\sigma_{xy},\sigma_{yz},\sigma_{zx}]^T$. The principal stress $(\sigma_1,\sigma_2,\sigma_3)$ are calculated from the stress components by the cubic equation:

$$\begin{bmatrix} \sigma_x-\sigma_0 & \sigma_{xy} & \sigma_{xz} \\ \sigma_{xy} & \sigma_y-\sigma_0 & \sigma_{yz} \\ \sigma_{xz} & \sigma_{yz} & \sigma_z-\sigma_0 \end{bmatrix} \tag{2}$$

where σ_0=principal stress (3 values)

The three principal stress are labeled $\sigma_1,\sigma_2,\sigma_3$. The principal stresses are ordered so that σ_1 is the most positive (tensile) and σ_3 is the most negative (compressive).

The von-Mises or equivalent stress σ_e is computed as:

$$\sigma_e=\left(\frac{1}{2}\left[(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2\right]\right)^{\frac{1}{2}} \tag{3}$$

2.3 Analysis Procedures

ANSYS/Workbench is performed through following steps:

1. Import or design model using Design Modeler.
2. Specify material properties.
3. Mesh the sub-model, define restriction and load.
4. Solve using Design simulation
5. Post-process
6. Evaluate and modify design repeatedly for better result.

3. Modeling and boundary conditions of reacting unit[3]

A reacting unit is designed for the polymerization reaction of pre-polymer. It is composed of two tanks and three shafts. Pre-polymer is polymerized in the inner tank. Between the inner and outer tank, there is a water-bath system, which is working as a temperature controlled medium for keeping the required temperature which pre-polymer need during the polymerization. Three shafts are used to support the unit stably during reaction process. Fig1. shows a schematic view of the designed reacting unit and its mesh model in the ANSYS/Workbench. Total mesh element numbers are 68101. Stainless steel is simulated as the material of reacting unit since it has strong resistance to corrosion, which will not react with the pre-polymer easily during polymerization process. Table 1. represents detailed specification of the reacting unit.

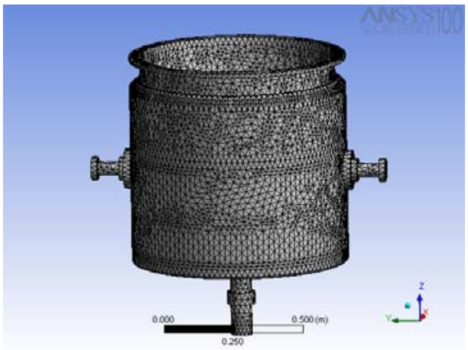

Fig. 1 Designed reacting unit and its mesh model

Table 1 Detailed specification of reacting unit

Mass	191.53 kg
Type	stainless steel
Young's Modulus	200 (MPa)
Poisson's Ration	0.3
Density	7850 kg/m^3

300kg pre-polymers are simulated to be made in the inner tank in one time. The static fluid pressure is dependent on the depth of liquid. In this study, we assume the pressure on the inner tank is 300Pa. To keep the reacting unit stably during the reaction process, the shafts on the side edge are simulated as fixed support. Fig. 2 shows the schematic diagram of the boundary conditions of reacting unit.

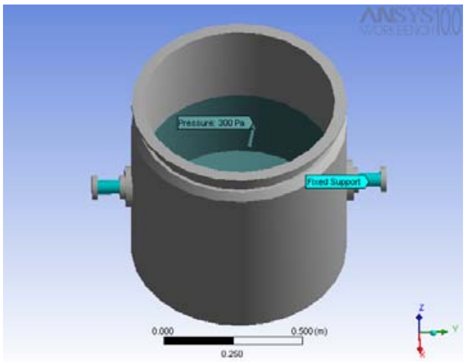

Fig. 2 Boundary conditions of reacting unit

4. Simulation results

4.1 Structural Analysis

Structural analysis of the polymerization reacting unit includes the simulation of Equivalent (von-Mises) stress and the total deformation. Analysis results are listed in Fig.3, Fig.4 and Table 2. The maximum value of von-Mises stress is 434.1 KPa, which occurs on upper edge of the funnel-shaped inner tank. The maximum value of the total deformation is 0.00287mm, which occurs at the same place that has maximum value of von-Mises stress. From the simulation results, it can be deduced that during the design of reacting unit, adding the thickness of the inner tank is required. It is also needed to improve geometric design of the funnel-shaped place in the inner tank.

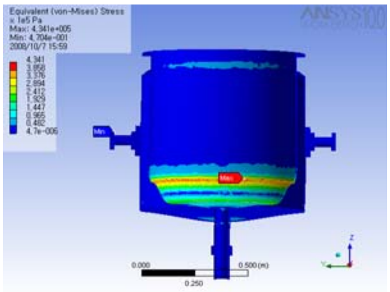

Fig. 3 Equivalent (von-Mises) stress

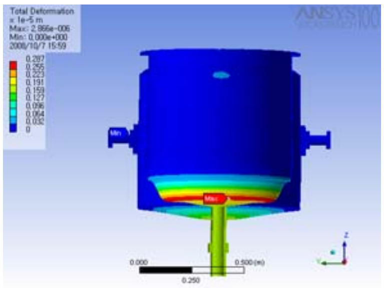

Fig. 4 Total deformation

Table 2 Structural analysis results

Type Value	(Maximum)
Equivalent (von-Mises) stress	434.1 KPa
Total deformation	0.00287mm

4.2 Mode Analysis

We get 6 series of frequency mode in range by performing mode analysis of the polymerization reacting unit. The value of frequency mode in range are listed in Table 3.

Table 3 Frequency mode in range

Frequency mode in range (Hz)					
1^{st}	2^{nd}	3^{rd}	4^{th}	5^{th}	6^{th}
93.5496	102.763	172.605	181.785	212.759	237.693

Harmonic analysis is used to determine the steady-state response of a linear structure to loads that vary sinusoidally (harmonically) with time. The results encompass the structure's response at several frequencies and provide graphs as a response quantity. In this study, according to the results of frequency mode in range, we set range Maximum at 300Hz and solution intervals at 10Hz. Fig. 5 shows the schematic diagram of the frequency response. It can be seen that when the frequency is 181Hz, the amplitude is highest. It is deduced the resonance is most likely to happen at this point with the reacting unit being destroyed. Fig.6 shows the 5^{th} frequency mode in range which occurs at 181Hz. The deformation of the reacting unit is more serious than others. Fig.7 shows the direction deformation of the reacting unit at 181Hz. The maximum value of direction deformation occurs on the wall of both inner and out tanks. Same as the structural analysis, the thickness of the tanks should be added for better geometric design of the reacting unit.

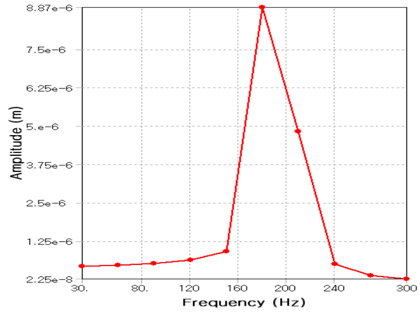

Fig. 5 Frequency response

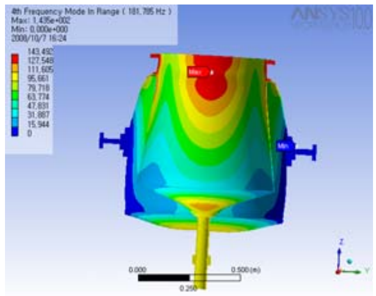

Fig. 6 5^{th} Frequency mode in range (Hz)

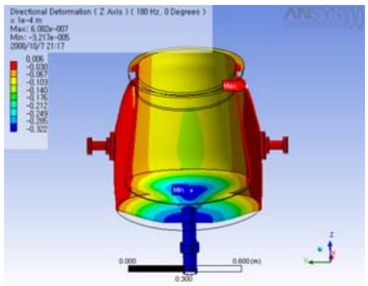

Fig. 7 Directional deformation at 181Hz

5. Conclusion

In this study, ANSYS/Workbench is used to simulate structural and vibration analysis of the polymerization reacting unit. The conclusions can be obtained as follows.

1.The maximum value of von-Mises stress and total deformation are 434.1 KPa and 0.00287mm, respectively. Both of them occur on upper edge of the funnel-shaped inner tank.

2.When the frequency is 181Hz, the amplitude is highest. It is deduced that the resonance is most likely to happen at this point with the reacting unit being destroyed.

Acknowledgment

This research was financially supported by the Ministry of Education, Science Technology (MEST) and Korea Industrial Technology Foundation (KOTEF) through the Human Resource Training Project for Regional Innovation No.(20070130134117).

Reference

1. "Fast-setting casting tape", United States Patent 4655208 http://www.freepatentsonline.com/4655208.html
2. "Theory Reference for ANSYS and ANSYS Workbench", http://www.kxcad.net/ansys/ANSYS/ansyshelp/theory_toc.html
3. "Introduction of ANSYS/Workbench", Printed by TAE SUNG SOFTWARE & ENGINEERING, INC.