SUSTAINABLE TECHNOLOGY FOR IMPROVING AUTOCLAVE PERFORMANCE USING FINITE ELEMENT APPLICATION

Willyanto Anggono[1], Ian Hardianto Siahaan[2], Andree Kadarna Tirta[3], and Satria Arief Budi[4]

Mechanical Engineering Department Petra Christian University[1,2,3,4]

Product Innovation and Development Centre Petra Christian University[1, 2, 3, 4]

Jl. Siwalankerto 121-131, Surabaya 60236

E-mail: willy@petra.ac.id[1]

Abstract

Sustainable technology is a part of broader concept of sustainable product development and sustainable development is an integral concept for achieving quality of limits, interdependence, fundamentals and equity. Autoclave is a strong, pressurized, steam-heated vessel, as for laboratory experiments, sterilization, or steaming. In general, this vessel contains of a view parts, they are the body which is the cylinder part of the tank, the heads which are used as the closure of the tank, the nozzle which are used as the fluid inlet and outlet port, and the saddle which are use to support body/tank of autoclave. The existing condition in this research, autoclave used to steam animal food materials and operated at 4 Bars pressured. The locking mechanisms are using 4 bolts to lock the door of the autoclave. The design of the autoclave can be operated up to 7 Bars. The bigger pressure operating condition, the faster processing happened.

Based on the existing condition, the goal of this research are finding and solving leakage problem in the area between door (head) and cylindrical shell when the autoclave operates at the pressure more than 4 Bars. The leakage starts when this autoclave operated at 4 Bars pressured. The bigger leakage happened when the autoclave operates at the bigger pressure (7 Bars).

ANSYS software based on the finite element method is used in this research. Based on the simulation result after re-design obtained, this autoclave can operates safely without leakage up to 7 Bars. Designing the mechanical product using ANSYS software can reduce materials, cost and time of the product development. ANSYS software based on finite element method is a sustainable technology for sustainable product development because simulation technology using ANSYS software can perform the performance of the machine during the design phase and reduce materials, cost and time of the product development.

Keywords: Sustainable Technology, Autoclave, Finite Element Method.

1. Introduction

Sustainable technology is a part of the broader concept of sustainable product development (Weenen, 2002) and virtual reality means existing or resulting in essence or effect though not in actual fact, form, or name (Anggono, 2006). Understanding the mechanical design is very hard to do because it is very hard to visualize the mechanical design product and the performance of the mechanical product during design phase.

Product design is always interesting to most of the people. It is also becoming more and more popular now a days. To predict the performance of the mechanical product during design phase is possible to do using Finite Element Method technology (ANSYS Software). Virtual reality technology using ANSYS software can perform the performance of the mechanical product during the design phase (Anggono, 2004).

Autoclave is a strong, pressurized, steam-heated vessel, as for laboratory experiments, sterilization, or steaming. In general, this vessel contains of a view parts, they are the body which is the cylinder part of the tank, the heads which are used as the closure of the tank, the nozzle which are used as the fluid inlet and outlet port, and the saddle which are use to support body/tank of autoclave. The existing condition in this research, autoclave used to steam animal food materials and operated at 4 Bars pressured. The locking mechanisms are using 4 bolts to lock the door of the autoclave. The design of the autoclave can be operated up to 7 Bars. The bigger pressure operating condition, the faster processing happened.

Figure 1. Autoclave using four key bolts

Based on the existing condition, the goal of this research are finding and solving leakage problem in the area between door (head) and cylindrical shell when the autoclave operates at the pressure more than 4 Bars. The leakage starts when this autoclave operated at 4 Bars pressured. The bigger leakage happened when the autoclave operates at the bigger pressure (7 Bars).

2. Theoretical Background

The Finite Element Method (FEM), also called FEA (Finite Element Analysis), is actually an approximate mathematical method for solving problems which can be determined by differential equations.

The main idea of FEM is to break a complicated problem with irregular edge conditions into small pieces (elements) of a finite size. Each piece is considered to be part of the main problem, thus connected to the other pieces via the global state information (i.e. deformation) of the element nodes, which are common nodes with the neighborhood elements.

For the small element itself, the internal physical laws (i.e. Hooks law for elastic deformation problems) can be calculated. The global problem can be transformed into a matrix of simple element equations which are connected by the condition, that common nodes undergo the same change of global state. Forces which act on the edge of the global thing can be simplified as acting at discreet nodes. This all together gives a big system of mostly linear equations which can be easily solved by computer.

The result is the change of global state for each node (i.e. the new node coordinates after the deforming of structures. Having this, further information for the small elements itself can be obtained, i.e. the element stresses in each direction (Budynas, 1999).

In general, autoclave contains of a view parts, they are the body which is the cylinder part of the tank, the heads cover which are used as the closure of the tank, the nozzle which are used as the fluid inlet and outlet port and the saddle which are used for support the autoclave body.

Working machine parts in general are subjected to multi axial loading, resulting in multi axial stress and strain. To utilize the data derived from one-dimensional magnitudes. We denote the effective stress and strain as effective stress and effective strain, respectively.

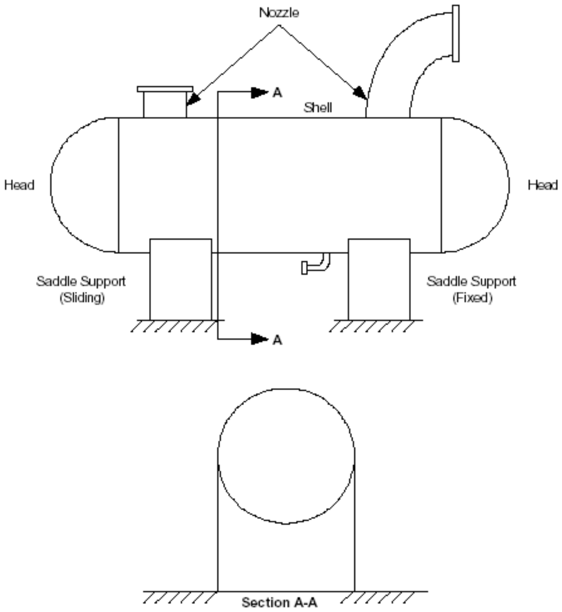

Figure 2. Autoclave terminology

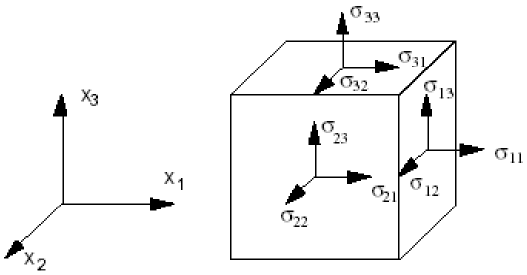

Figure 3. Three dimension stress diagram.

There are several known methods to derive effective stress and strain. Those presented here are based on von Mises theory and were chosen for their applicability to machine design. In the material failure by von Mises theory, failure by yielding occurs when, at any point in the body, the elastic strain energy absorbed by the unit volume as a result of its change in shape, distortion energy by the combined stress becomes equal to that associated with yielding in a simple tension test.

The complex state of stress gets more or less compared with a uni-axial stress.

Von Mises effective stress:

$$((\sigma_{11}-\sigma_{22})^2 + (\sigma_{22}-\sigma_{33})^2 + (\sigma_{33}-\sigma_{11})^2 + 6(\sigma_{12}^2 + \sigma_{23}^2 + \sigma_{13}^2))^{1/2} \quad (1)$$

Von Mises effective strain is defined by:

$$((\varepsilon_1+\varepsilon_2)^2 + (\varepsilon_2+\varepsilon_3)^2 + (\varepsilon_3+\varepsilon_1)^2)^{1/2} / ((1+\upsilon)\, 2^{1/2}) \quad (2)$$

To be familiar with some element types and potential element types to fulfill the needs in this problem that can be used in ANSYS software are needed before using the ANSYS program to solve the problem. The suitable element type that can be used to solve the problem is SOLID95 3-D 20-Node Structural Solid (ANSYS Inc., 2002).

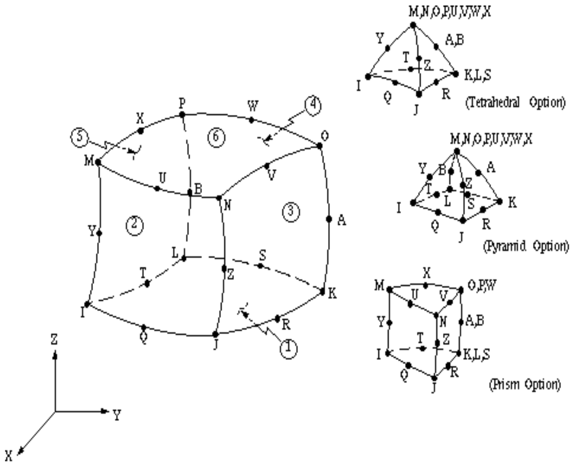

Figure 4. SOLID95 element type 3-D 20-node structural solid

SOLID95 can tolerate irregular shapes without as much loss of accuracy. SOLID95 elements have compatible displacement shapes and are well suited to model curved boundaries. The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. The element may have any spatial orientation. The element has plasticity, creep, stress stiffening, large deflection, and large strain capabilities.

3. Research Methodology

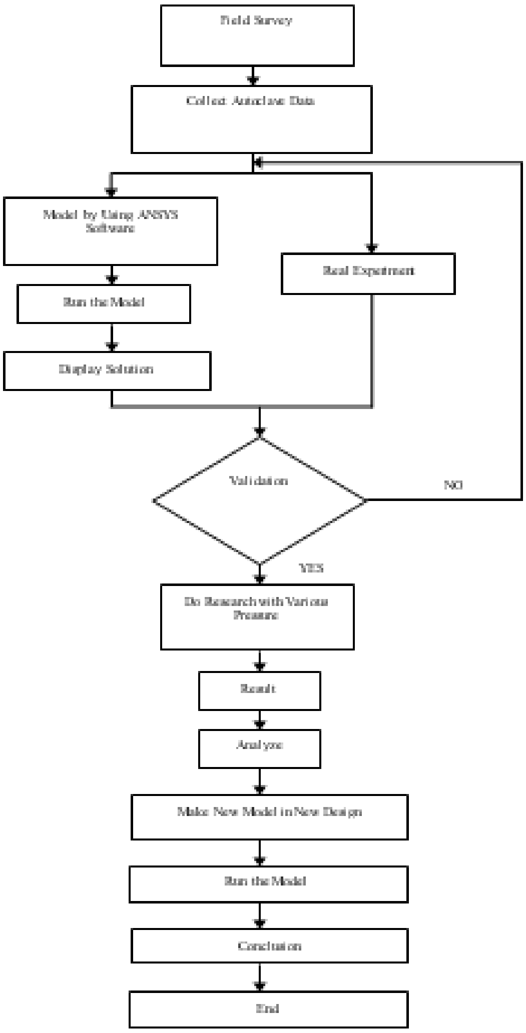

Figure 5. Research methodology

4. Result and Discussion

The simulation is very important to solve the problem. The simulation is performed using half model of the product because the autoclave is a symmetry product as well and the symmetry feature in ANSYS software can perform the symmetry product using half model. The meshing model for head cover autoclave is shown in figure 6.

After meshing process has finished, then the internal pressure given to the cover head from the 1 Bar until 4 Bars.

Displacement which happened to the head cover is shown in figure 7.

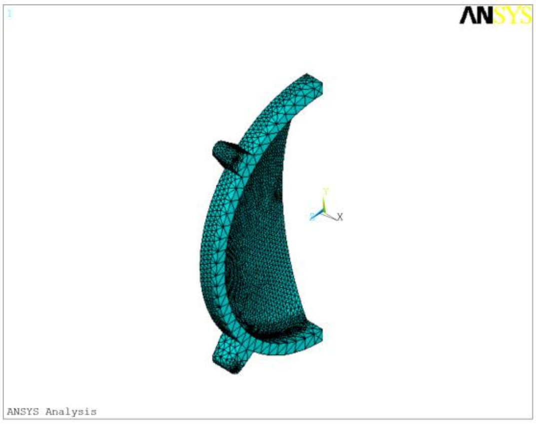

Figure 6. Meshing for the head cover of the model

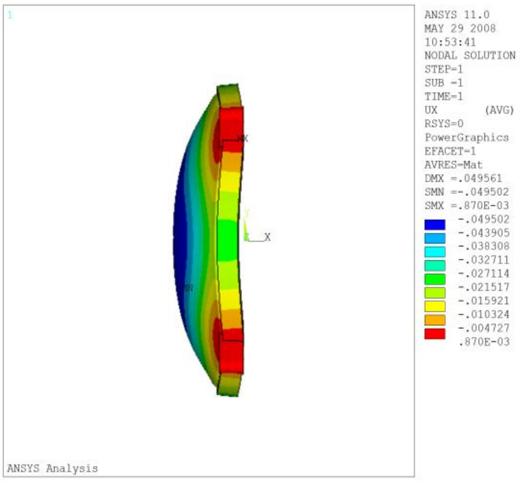

Figure 7. Displacement at the head cover

Based on the simulation result (figure 7), the maximum displacement happened in the middle above the key bolts. The displacement which happened is shown in the 10 nodes observation that taken on the along side section of the cover as shown in the figure 8 and table 1.

Using the same procedure is doing to the body tank model, which symmetric shell used to the body. The meshing for body tank can be seen in figure 9.

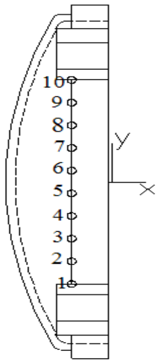

Figure 8. Ten nodes observation at the head cover

Table 1. Displacement which happened at the 10 nodes when apply pressure1 Bar until 4 Bars.

No	Node Number	Displacement (mm)			
		At the 1 bar	At the 2 bar	At the 3 bar	At the 4 bar
1	54999	0.0021184	0.0042369	0.0063553	0.0084737
2	54986	0.0078903	0.015781	0.023671	0.031561
3	54785	0.013483	0.026965	0.040448	0.053931
4	54715	0.018023	0.036046	0.054069	0.072093
5	54823	0.020344	0.040689	0.061033	0.081378
6	1668	0.020344	0.040689	0.061033	0.081378
7	1693	0.018023	0.036046	0.054069	0.072093
8	1720	0.013483	0.026965	0.040448	0.053931
9	1747	0.0078903	0.015781	0.023671	0.031561
10	1778	0.0021184	0.0042369	0.0063553	0.0084737

Figure 9. Meshing for the body tank

After meshing process has finished, then the pressure is given to the body tank from 1 Bar until 4 Bars too. Displacement which happened to the body tank is shown in figure 10.

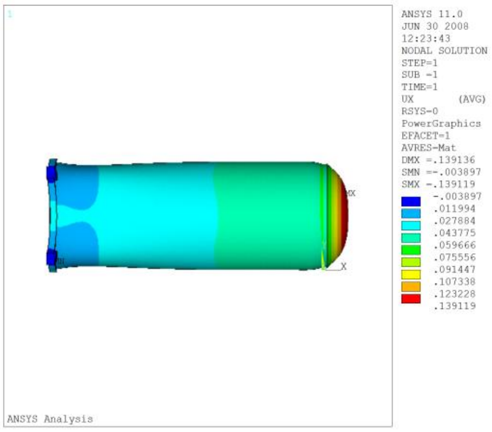

Figure 10. Displacement at the body tank

As shown in figure 10, displacement happened in the side section of the body when pressure was given from 1 Bar until 4 Bars is shown in figure 11 and table 2.

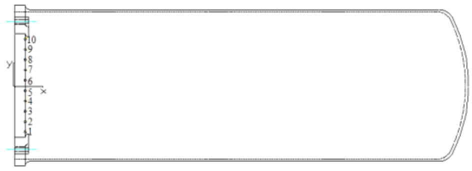

Figure 11. Ten nodes observation at the body tank

Table 2. Displacement which happened at the 10 nodes when apply pressure1 Bar until 4 Bars.

No	Node Number	Displacement (mm)			
		At the 1 bar	At the 2 bar	At the 3 bar	At the 4 bar
1	2727	0.001086	0.021732	0.032598	0.043464
2	2716	0.005798	0.035971	0.053956	0.071941
3	2706	0.023147	0.046294	0.069442	0.092589
4	2696	0.026610	0.053220	0.079830	0.10644
5	2686	0.028420	0.056840	0.085260	0.11368
6	73297	0.028420	0.056840	0.085260	0.11368
7	73287	0.026610	0.053320	0.079830	0.10644
8	73279	0.023147	0.046294	0.069442	0.092589
9	73269	0.005798	0.035971	0.053956	0.071941
10	73260	0.001086	0.021732	0.032598	0.043464

The total displacement value that happened in the all autoclave can be calculated by added displacement value which happened in the head cover and displacement value which happened in the body tank, as shown in figure 12 and table 3.

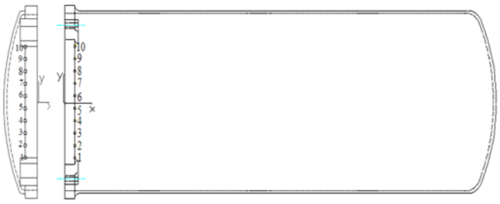

Figure 12. Total displacement at the autoclave

Table 3. Total displacement which happened in the 10 node when apply pressure 1 Bar until 4 Bars.

No	*Displacement* (mm)			
	At the 1 bar	At the 2 bar	At the 3 bar	At the 4 bar
1	0.0032044	0.0259689	0.0389533	0.0519377
2	0.0136883	0.051752	0.077627	0.103502
3	0.03663	0.073259	0.10989	0.14652
4	0.044633	0.089266	0.133899	0.17853
5	0.048764	0.097529	0.146293	0.195058
6	0.048764	0.097529	0.146293	0.195058
7	0.044633	0.089266	0.133899	0.17853
8	0.03663	0.073259	0.10989	0.14652
9	0.0136883	0.051752	0.077627	0.103502
10	0.0032044	0.0259689	0.0389533	0.0519377

Validation is used to compare the displacement between real experiment result and the result of ANSYS simulation result. Real experiment activity is shown in figure 13.

Working pressure of the autoclave can be seen in pressure gauge which placed behind autoclave body (figure 14).

Figure 13. Real experiment activity

Figure 14. Pressure gauge

Table 4. Comparison between ANSYS result and real experiment

No	*Displacement (mm)*			
	Simulation			*Real*
	Tutup	Bodi	Total (tutup + bodi)	*Experiment*
1	0.0084737	0.043464	0.0519377	0.054
2	0.031561	0.071941	0.103502	0.105
3	0.053931	0.092589	0.14652	0.149
4	0.072093	0.10644	0.17853	0.179
5	0.081378	0.11368	0.195058	0.198
6	0.081378	0.11368	0.195058	0.198
7	0.072093	0.10644	0.17853	0.179
8	0.053931	0.092589	0.14652	0.149
9	0.031561	0.071941	0.103502	0.105
10	0.0084737	0.043464	0.0519377	0.054

The result from ANSYS software simulation and real experiment can be seen in table 4 and the accuracy of ANSYS software simulation is good compare with real experiment (figure 15).

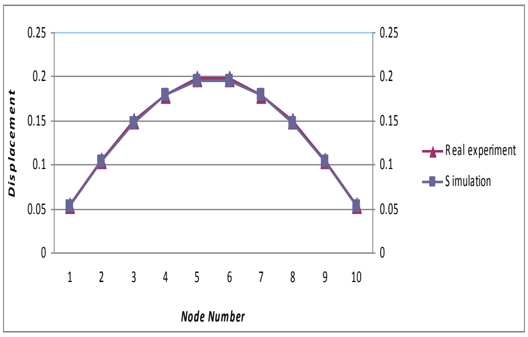

Figure 15. New design of the head cover

After validations process is done from the ANSYS simulation result and the real experiment result, can be known that the reason leakage happened in the side autoclave design. The next step, create a new design where in the new design is not happened leakage again like which happened in the old design when the autoclave was operated in pressure 7 Bars. The new design can we see in figure 16 and figure 17.

Figure 16. New design of the head cover

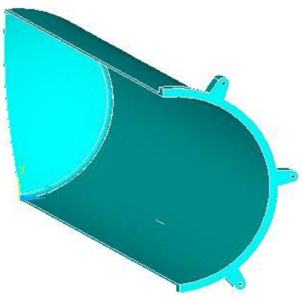

Figure 17. New design of the body tank

In the new design, the same procedure with the old design which symmetric shell used to the head cover and the body tank. The meshing for the head cover can be seen in figure 18.

Figure 18. Meshing for the head cover new design

After meshing process has finished then the pressure is given to the head cover at 7 Bars. Displacement which happened can be seen in figure 19.

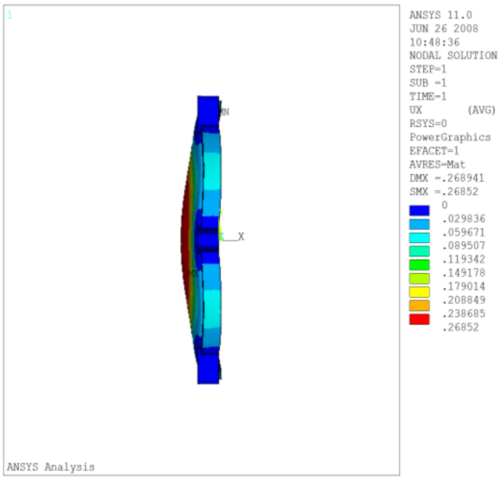

Figure 19. Displacement in the head cover

Using the same procedure is doing to the body tank model, which symmetric shell used to the body. The meshing for body tank can be seen in figure 20.
After meshing process has finished then the pressure is given to the head cover at 7 bars. Displacement which happened can we see in figure 21.

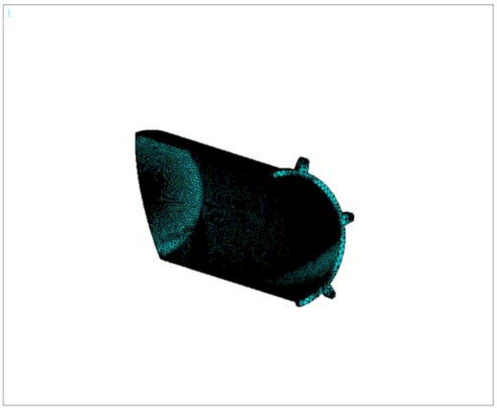

Figure 20. Meshing the body tank

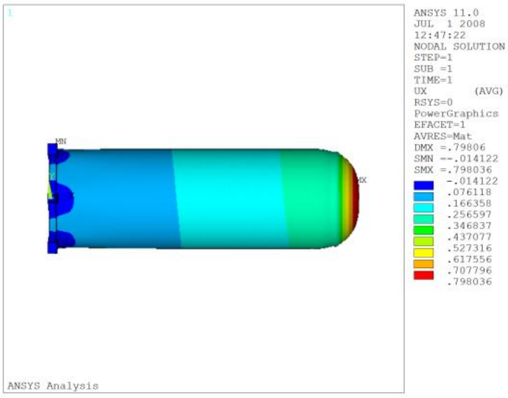

Figure 21. Displacement in the body tank

The total displacement value that happened in the all autoclave can be calculated by added displacement value which happened in the head cover and displacement value which happened in the body tank can be seen in figure 22 and table 5.

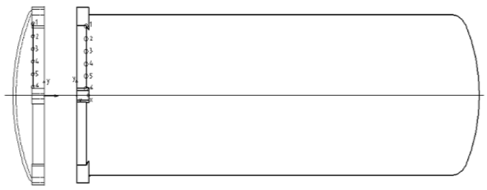

Figure 22. Total displacement at 6 nodes

From table 5 can be seen that the displacement value that happened in the side section of the autoclave new design (0.120237 mm) become smaller than displacement value that happened in the autoclave old design (0.195058 mm).

So in the autoclave new design can operated in pressure 7 Bars and it is safe.

Table 5. Displacement total which happened in the 6 nodes in the new design autoclave. (At working pressure 7 Bars)

No	*Displacement* (mm)		
	Head Cover	Body Tank	Total
1	0.0023790	0.019527	0.021906
2	0.0050861	0.065623	0.070709
3	0.013483	0.089358	0.102841
4	0.023861	0.096376	0.120237
5	0.0068903	0.079001	0.0858913
6	0.0016734	0.022609	0.0242824

Conclusions

The full scale tests (real experiment test) are expensive and require a lot of time, material and money. The full scale tests is not the Sustainable Product Development way of autoclave design. Using Finite Element Method Technology changing material and shape are very easy to do and many other designs can be made easily. Reducing cost, material and time of the design is the most important aspect. Reducing cost, material and time of the mechanical design using virtual reality is a sustainable product development in the mechanical design

Based on the simulation result after re-design obtained, this autoclave can operates safely without leakage up to 7 Bars. Designing the mechanical product using ANSYS software can reduce materials, cost and time of the product development. ANSYS software based on finite element method is a sustainable technology for sustainable product development because simulation technology using ANSYS software can perform the performance of the machine during the design phase and reduce materials, cost and time of the product development.

References

[1]. Anggono, Willyanto. (2004) Peningkatan Unjuk Kerja Desain Flexible Shield untuk Pompa Sabun dengan Menggunakan Metode Elemen Hingga, Jurnal Teknik Mesin, 6, 57-64.

[2]. Anggono, W. (2006) Analisa Pengaruh Radius Heads Terhadap Besar Tegangan Maksimum pada Air Receiver Tank Horisontal dengan Menggunakan Metode Elemen Hingga. Proc. National Seminar on Application and Research in Industrial Technology 2006 (Yogyakarta) April 27, pp 78-86.

[3]. ANSYS, Inc. (2002) *ANSYS 7.0 Documentation*, USA.

[4]. Budinski, K.G. and Budinski M.K. (2002) Engineering Materials Properties and Selection, Prentice Hall,USA pp 3-67.

[5]. Budynas, Richard G. (1999). Advanced strength and applied stress analysis. Singapore: McGraw-Hill Book Company.

[6]. Msc Software Corporation (2003) *Non Linear Finite Element Analysis of Elastomers*, USA.

[7]. Pugh, S. (1991). Total Design: Integrated Methods for Successful Product Engineering, Addison-Wesley Publishing Company, Inc., USA pp 8-59.

[8]. Pugh, S. (1996). Creating Innovative Products Using Total Design, Addison-Wesley Publishing Company, Inc., USA pp 7-61.

[9]. Tirta, Andree K. (2008). Perencanaan *Autoclave* dengan menggunakan aplikasi metode elemen hingga, Tugas Akhir S1 Jurusan Teknik Mesin Universitas Kristen Petra.

[10]. Ullman, David G. (2003). The mechanical design process. New York: McGraw-Hill Book Company.

[11]. Weenen, J C van. (2002) Concept, context, and co-operation for sustainable technology. Proc. International Seminar on Design and Manufacture for sustainable development 2002 (Liverpool) June 27-28, pp 3-12.