Journal of
Ship&Ocean
Technology
©DECEMBER 2008

SOTECH Vol. 12, No. 4, pp. 7 ~ 19, 2008

FE Model Based Parametric Study Support System

Beom-Seon Jang[1]

[1] Seoul Offshore Design Center, Offshore Basic Engineering Team, Samsung Heavy Industries, Geoje, 656-710, Korea ;
Corresponding Author: beomseon.jang@samsung.com

Abstract

In preliminary ship design, a parametric study is a more realistic way to explore design space and analyze design problem than an optimization technique due to time-consuming computational work or a difficulty in incorporating all constraints into the optimization formulation. In the parametric study, feasible alternatives are examined in various aspects; the best one can be selected. Among the aspects, the strength assessment by FE analysis is an essential process in the ship design. This paper proposes a system to facilitate a parametric study for FE model based on design of experiment (DOE). It works on a FE pre-processor environment and assists a user to define a parametric study by interacting with FE model. It also provides an interface module with a FE solver in order to control the input file and extract predefined FE results from the output file. Based on the proposed system, a better understating and a better design are expected to be achieved.

Keywords: parametric study, FE analysis, design of experiment

1 Introduction

Design variables can be classified into continuous variables and discrete design variables. In actual ship design, most scantlings such as plate thickness or section profile of angle type stiffener are discrete variables due to their manufacturing standards. Other parameters related with structural arrangement such as stiffener spacing, web spacing or double bottom height are also expressed in tens or hundreds (mm) as a matter of convenience.

As an approach to treat these discrete variables in optimization, an optimal point obtained in continuous design space is replaced as a discrete point which is the closest to the optimal point. However, this approach requires additional searching process in order to find out the best among the neighboring discrete points satisfying constraints. Moreover, if the optimization requires time-consuming analyses, this approach may become more inefficient than an approach to optimize in discreet design space directly because it can be avoided an exhaustive and sophisticated search around an optimum until satisfying a convergence criterion.

For this reason, optimization techniques have been developed to handle discrete variables directly. Park and Lee (1995) suggested an interactive design optimization method and Salajegheh and Vanderplassts (1993) suggested a branch and bound method. Rajeev and Krishnamoorthy (1992) and Wu and Chow (1995) treated discrete design variables by employing genetic algorithm which is known to be able to treat discrete variables effectively as well as its capability of finding out global optimum point instead of sticking in local optimum (Goldberg 1989). However, since these algorithms are still based on an exhaustive exploration of design space requiring a lot of evaluations of constraints or objective function, it is not appropriate for an actual engineering design problem requiring complicated calculations such as finite element analysis.

Design of experiment (DOE) based approach has been known effective to a design problem where a physical experiment or a large amount of computational work prohibits a thorough exploration of design space. Among the DOE methods, orthogonal array is known to be very efficient in design space exploration and has been introduced in many fields (Ku et al 1998, Phadke 1989, Peace 1995).

A lot of efforts have been made in shape optimization field. The structural shape optimization has been studied based on well established shape design sensitivity analysis. Extensive reviews on structural shape optimization and shape sensitivity analysis can be found in Haftka and Grandhi (1986) and Kwak (1994). However, shape and topology optimizations are appropriate to small homogenous mechanical parts. It is practically impossible to apply the techniques to a conceptual design of large-sized structure such as a ship, an aero plane and a car due to the difficulties in representing their complex structure as design variables and formulating a lot of design restrictions in mathematical forms.

As another effort of shape optimization is to perform optimization in CAD environment based on parametric modeling technique and automatic mesh generation technique. Optimization module controls parameters defined in CAD geometry and FE model is automatically constructed for the CAD geometry. Then FE analysis is performed to evaluate design constraints or objective function. Those integrations are made in CAD systems using API (Application Programming Interface). DS/FDM (Shin and Kwak 2001) utilizes finite difference method to compute approximate derivatives required in gradient based optimization algorithm in Pro/Engineer system. DS/MEMS (Huh 2000) enables an optimization for MEMS structure considering complex fields in SolidWorks environment and DS//I-DEAS in I-DEAS system. Although those approaches have a limitation in changing geometry from its initial shape compared to topology optimization, it has much more flexibility in optimization formulation. An interface with FE analysis module enables to use any outputs from FE results for constraints or objective function.

However, there still exists a difficulty in applying the approach to a complex structure such as a ship structure composed of stiffened plates because it is not possible to automatically generate perfect mesh for the ship structure without additional manual mesh modification work during the optimization. It is another obstacle to construct complete parametric relations between curved surfaces and lines such as outer hull, longitudinal stiffeners on the hull or intersecting horizontal or transverse members.

Meanwhile, the variation of ship shape from structural point of view is restricted at the initial ship design stage. Main dimensions, and cargo tank sizes are fixed from the start of contract. Only limited parts such as depth of double bottom or side web breadth are allowed to be changed to improve torsional strength of large containership at the stage of assessing structural strength. For this reason, simple change by a simple parallel translation of specific nodes can cover the considered change of ship structure.

In this research, a system is developed to facilitate a parametric study based on DOE methods. It operates on a FE pre-processor (MSC.PATRAN) environment and assists a user to define parametric study with interacting with FE model. The parametric study includes a variation of scantlings and simple shape changes. It also provides a module to interact with FE solver (MSC.NASTRAN) by parsing its input and output files. A set of experiments can be analyzed using Analysis of Variance (ANOVA) table and the relations between parameters and responses are identified in graphs.

In Section 2, the difficulty in applying optimization to preliminary ship design problem and the necessity of parametric study are discussed. Section 3 explains the features of a proposed parametric support system and Section 4 describes an overall procedure of the system. An application example of full container ship is presented in Section 5 and conclusion is laid in Section 6.

2 Parametric study as an alternative to optimization in preliminary ship design

Ship structural design field has some difficulties in employing an FE analysis based optimization method due to the following reasons.

First, computational time for FE analysis hinders the application of optimization technique in structural design requiring FE analysis. Even if computational performance has been improved drastically, the demand on high accuracy of strength assessment has also grown consequently. Therefore, the iterative FE analysis still remains a burden in the application of optimization.

Second, the large number of constraints is another obstruction. All elements in FE model should satisfy allowable stress criteria and buckling criteria under at least tens of load cases. Moreover, if finite difference method is used for the constraints in a gradient based optimization technique, the number of FE analysis gets doubled or tripled. In direct search methods, more constraints are taken into account, more exhaustive exploration is needed for finding feasible design space.

Third, the optimization should take into account a lot of construction related constraints that are hard to be expressed in explicit mathematical formula. For example, arrangement of seam line, or block butt line should be considered in determining plate thickness. The difference between neighboring plate thicknesses should be below a certain value in order to avoid stress concentration caused by thickness transition. Those considerations are critical to obtain a practical optimal design although they are hard to be included in the formulation of optimization problem.

Fourth, ship preliminary design is characterized by its strong reference to previous design cases. Usually, commercial vessel design begins with the most similar existing design or a standard design, so called a parent ship. The selected original design is adapted to meet given specifications and classification regulations. Large change is not necessary or even not preferred sometimes because it is not proven through a real operation. It is also related with a conservative characteristic of ship design due to its huge financial and environmental damage in case of accidental oil leakage or sinking. Uncertainties in predicting structural behavior of a sailing vessel also contributes to the maintenance of the design practice to refer to proven vessels. This design practice discourages the introduction of optimization techniques.

Fifth, construction cost such as welding cost or the number of pieces in assembly process are critical to total cost as well as material cost in real ship design. However, it is difficult to define construction cost function explicitly. Therefore, it is a more suitable approach to select one of feasible alternatives after a qualitative examination from all aspects including the construction cost than to use an optimal design obtained from an optimization with an ambiguous definition of fabrication cost.

The above-mentioned reasons make designers remain in an iterative way to search a better design in actual preliminary ship design. This research adopts a design of experiment (DOE) for a systematic search as a good alternative to the heuristic search. The DOE is known to be an efficient way for exploring design space and understanding a design problem. As an object to be controlled, FE model is selected instead of geometry based product model since it can avoid a burden to generate FE mesh even if the information to be contained in FE model is less than the product model.

This paper proposes a system to support a parametric study using design of experiments with interfacing with FE model. The effects of parameters defined in FE model on FE analysis results are investigated in a systematic way and the best solution through the parametric study can be selected. In the following section, the overall framework of the system and main characteristics are described in detail.

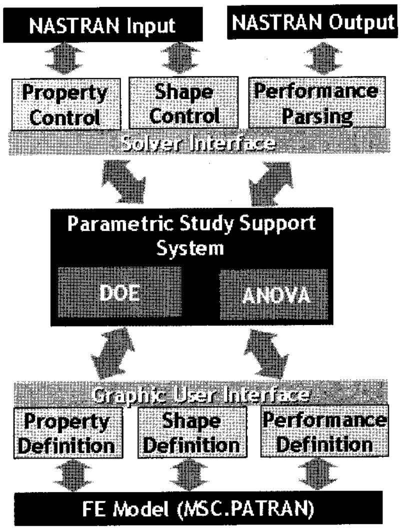


Figure 1: Interface with FE preprocessor and FE solver

3 DOE based parametric study support system

This research provides a parametric study support system working in FE pre-processor (MSC.PATRAN) environment. The access to the FE model and its control are enabled by PCL language provided by MSC.PATRAN. Overall framework of the system is illustrated in Figure 1.

The characteristics of the proposed system are described as follows.

First, all parameters and performance can be defined by a direct interface with FE model on a FE pre-processor (MSC.PATRAN) environment through graphic user interface (GUI). Plate thickness, stiffener section profile, material properties, or coordinate value for specific nodes. A simple shape change is enabled by a limited change of coordinate values of selected nodes can be chosen for design parameters. Everything from FE analysis can be defined as a response such as strain, displacement, strain energy and so on. Parameters can be defined by clicking or selecting some

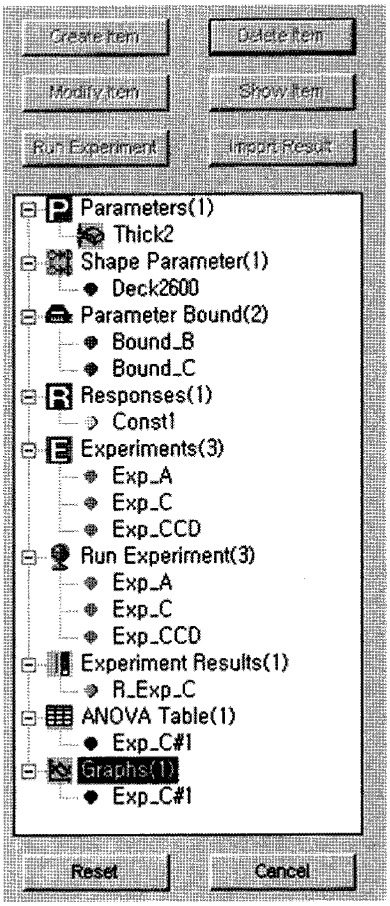


Figure 2: Main GUI menu in tree type

entities in a window displaying FE model. This is distinguished by a text-parsing approach employed in commercial optimization tools such as ISIGHT (1999), or VisualDOC (1998) where a user has to define design parameters or performances by designating a specific location of input and output files of FE solver. It is not convenient since it requires a complete understating of the input and output files. In the proposed system, all parameters and response variables are defined in user friendly GUI and a user doesn't have to know about the input/output file.

Second, all data related with the parametric study such as parameters, bounds for the parameters, experiment matrix, experimental results, ANOVA table, graphs and so on can be stored in a FE model database file. User defined data block provided in MSC.PATRAN data base can be used for the storage. Therefore, no other file is generated for the parametric study except the FE model data base file. This facilitates the management of data files.

Third, work process is defined in tree type GUI as depicted in Figure 2. A user can be guided by just following items listed in the tree type GUI. Already defined data can be also accessed directly by clicking the corresponding item in the tree. Since MSC.PATRAN doesn't support left mouse button, six buttons located at the upper part of the GUI are designed to play a role of the left mouse button. The number of buttons and their titles are changed depending on the currently activated item.

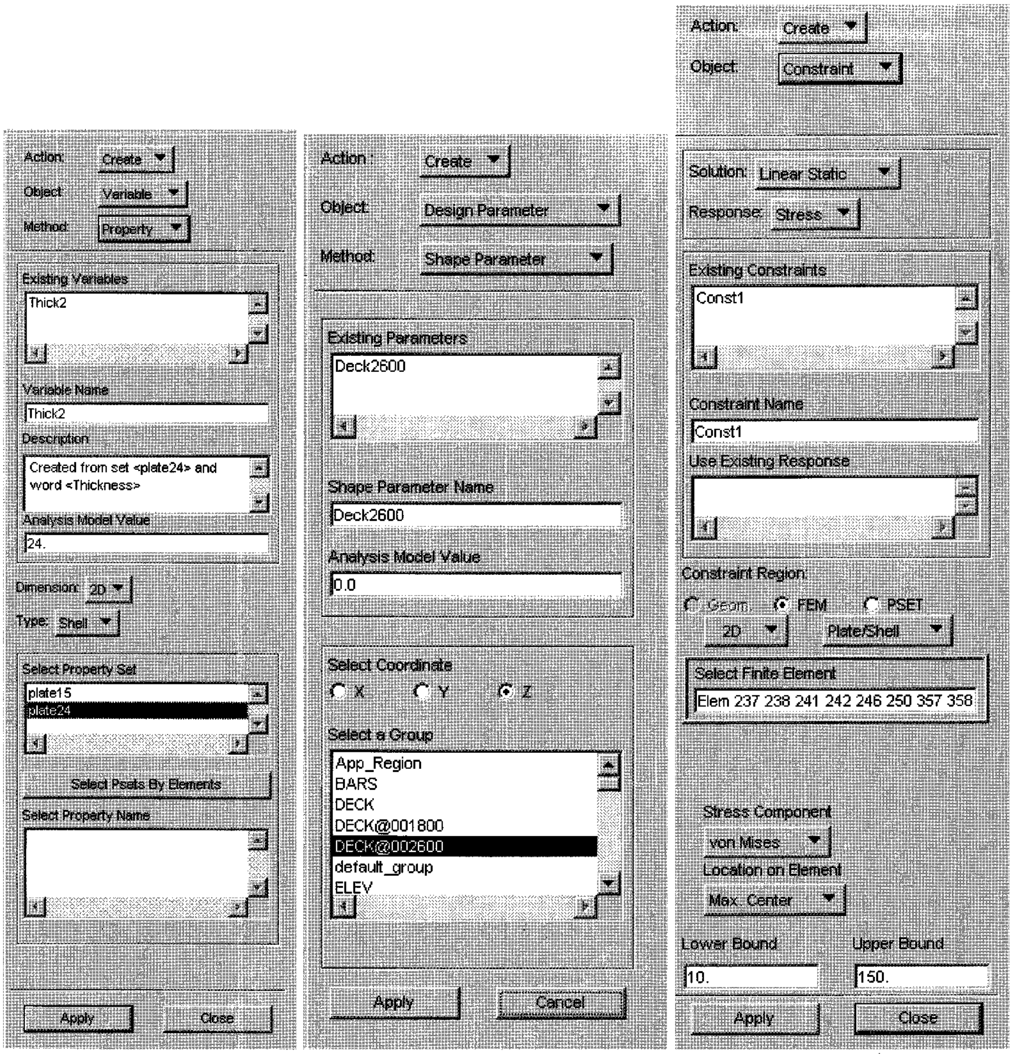
In the next section, whole procedure of the system is described step by step.

4 Procedures of parametric study

The system consists of total 8 steps and they are explained in detail one by one.

Step 1: Define property parameters. (see Figure 3 (a))
The following properties can be defined as parameters and current value can be assigned in this window.

- Area of rod element
- Plate thickness of shell element
- Section profile of beam element – ex) breath, height, thickness of web, thickness of flange of a stiffener
- Material properties – density, elastic modulus, Poisson ratio and so on


Action: Create
Object: Variable
Method: Property
Existing Variables: Thick2
Variable Name: Thick2
Description: Created from set <plate24> and word <Thickness>
Analysis Model Value: 24.
Dimension: 2D
Type: Shell
Select Property Set: plate15, plate24
Select Property Name:

Action: Create
Object: Design Parameter
Method: Shape Parameter
Existing Parameters: Deck2600
Shape Parameter Name: Deck2600
Analysis Model Value: 0.0
Select Coordinate: ○ X ○ Y ● Z
Select a Group: App_Region, BARS, DECK, DECK@001800, DECK@002600, default_group, ELEV

Action: Create
Object: Constraint
Solution: Linear Static
Response: Stress
Existing Constraints: Const1
Constraint Name: Const1
Use Existing Response:
Constraint Region: ○ Geom ● FEM ○ PSET
2D
Plate/Shell
Select Finite Element: Elem 237 238 241 242 246 250 357 358
Stress Component: von Mises
Location on Element: Max. Center
Lower Bound: 10.
Upper Bound: 150.

(a) (b) (c)

Figure 3: GUI examples for defining (a) property parameter (b) shape parameter (c) response.

Step 2 : Define shape parameters (see Figure 3 (b))

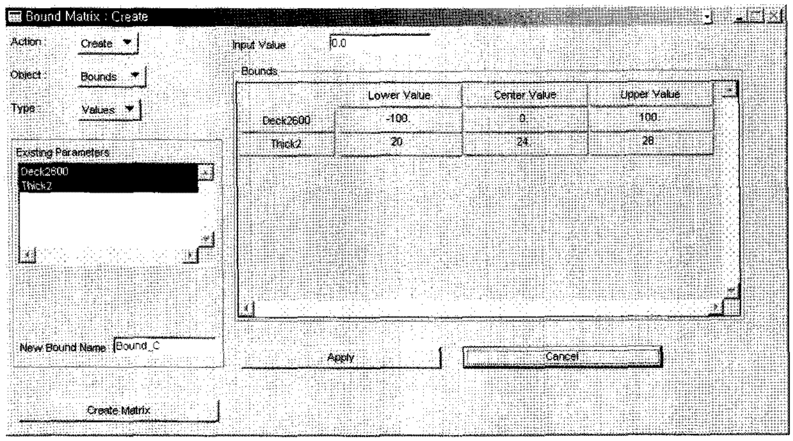
- A shape parameter can be defined by selecting a coordinate axis and a group containing only nodes to be translated.
- Shape can be changed by modifying the coordinate value of nodes in the selected group. For example, double bottom height can be varied by shifting nodes on inner bottom in z direction
- Only slight change of nodes is permitted in order not to spoil mesh connectivity. The shape change is suitable to FE model composed of coarse mesh of which size is three or four times of longitudinal stiffeners such as full ship FE model used at the early design stage. Since bottom girders placed between inner bottom and bottom plate consist of one element, shape change doesn't affect element connectivity but just modifies the size of shell elements.

Step 3 : Define response to be investigated (see Figure 3 (c))

- All FE analysis results such as displacement, stress, strain, strain energy, or grid point force for specific nodes or elements can be chosen as a response. Total weight can be also easily calculated from the FE model.

Step 4 : Define parameter bounds (see Figure 4)

- Define the bounds of parameters defined in Step 1 and 2
- The bounds are used for determining actual parameter levels along with an experiment matrix to be defined at the next step.



Action : Create
Object : Bounds
Type : Values
Input Value 0.0

Existing Parameters
Deck2600
Thick2

New Bound Name Bound_C

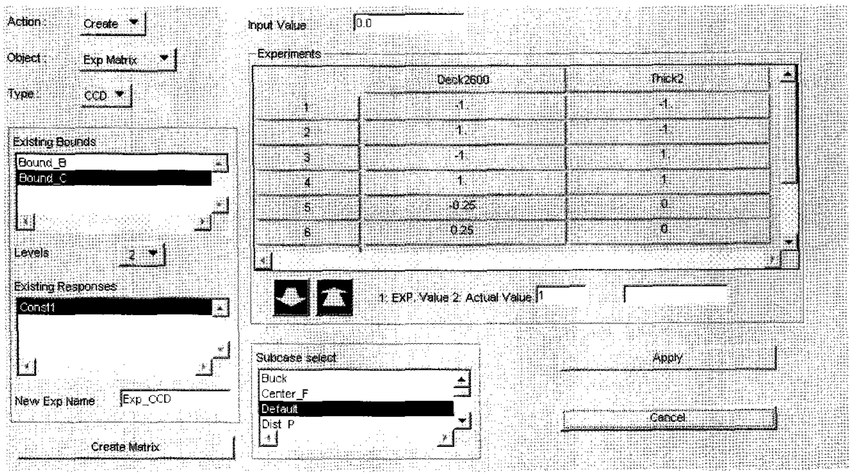
Bounds

	Lower Value	Center Value	Upper Value
Deck2600	-100.	0.	100.
Thick2	20	24	28

Figure 4: A GUI example for defining parameter bounds.

Step 5 : Define an experiment matrix (see Figure 5)

- Define an experiment matrix to contain a set of combinations of parameter levels. This system provides an automatic generation of the matrix using central composite design and full factorial design.
- The matrix can be also created or edited manually by filling up matrix manually.



Action : Create
Object : Exp Matrix
Type : CCD
Input Value 0.0

Existing Bounds
Bound_B
Bound_C

Levels 2

Existing Responses
Const1

New Exp Name Exp_CCD

Experiments

	Deck2600	Thick2
1	-1.	-1.
2	1.	-1.
3	-1.	1.
4	1.	1.
5	-0.25	0.
6	0.25	0.

1: EXP. Value 2: Actual Value 1

Subcase select
Buck
Center_F
Default
Dist_P

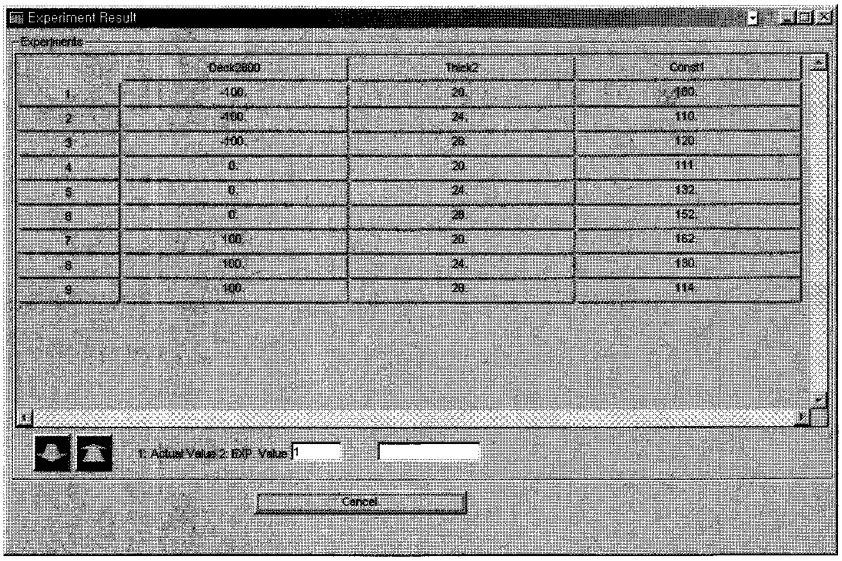
Figure 5: A GUI example for defining experiment matrix.

Step 6 : Execute a set of experiments

- An input file of FE solver (MSC.NASRTAN) and an input file to contain the information of the defined parametric study are generated automatically. The input file is used for an execution module to perform the parametric study.
- The execution module repeats FE analysis as varying specific parts of the FE solver input file as planned.
- The module also extracts the pre-defined response by parsing NASTRAN output text file.
- If analysis is completed, a file to contain the experiment results is generated.
- A user only has to run the execution module and all processes are performed automatically.

Step 7 : Import experimental results (see Figure 6)

- The experiment result file is imported to the system.
- The results are summarized in a table.



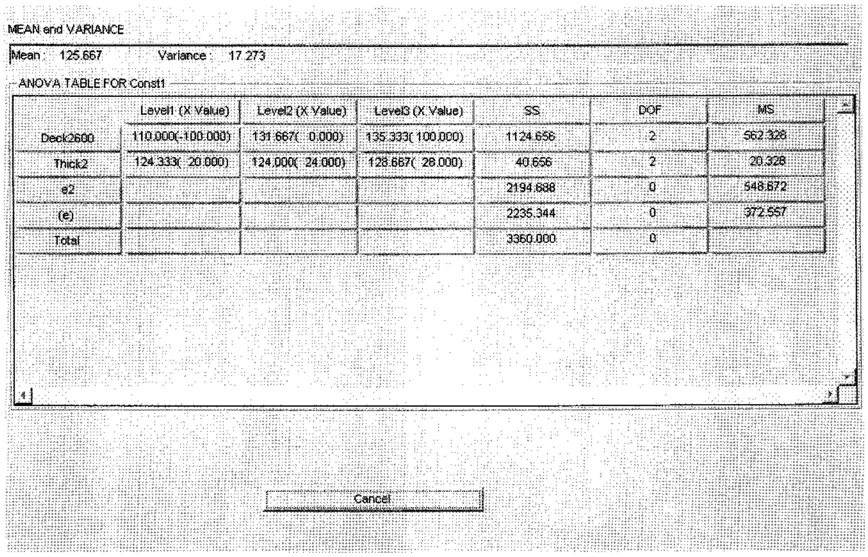
	Deck2600	Thick2	Const1
1	-100.	20.	100.
2	-100.	24.	110.
3	-100.	28.	120.
4	0.	20.	111.
5	0.	24.	132.
6	0.	28.	152.
7	100.	20.	162.
8	100.	24.	130.
9	100.	28.	114.

1: Actual Value 2: EXP Value 1

Figure 6: A GUI example to show imported experiment results

Step 8: ANOVA Table. (see Figure 7)

- ANOVA (Analysis of Variance) table is provided through a statistical analysis for the experimental results.
- From this analysis, the effect of parameters on the specific response can be quantified and their relations can be identified.



MEAN and VARIANCE

Mean : 125.667 Variance : 17.273

ANOVA TABLE FOR Const1

	Level1 (X Value)	Level2 (X Value)	Level3 (X Value)	SS	DOF	MS
Deck2600	110.000(-100.000)	131.667(0.000)	135.333(100.000)	1124.656	2	562.328
Thick2	124.333(20.000)	124.000(24.000)	128.667(28.000)	40.656	2	20.328
e2				2194.688	0	548.672
(e)				2235.344	0	372.557
Total				3360.000	0	

Figure 7: A GUI example to show ANOVA Table.

Step 9 : Interaction plot for response (see Figure 8)

- Using the data in ANOVA table, main effects and interaction effects of parameters on a response can be plotted. This plot presents the relations between parameters and a response in a manner that is easier to comprehend than the ANOVA table.

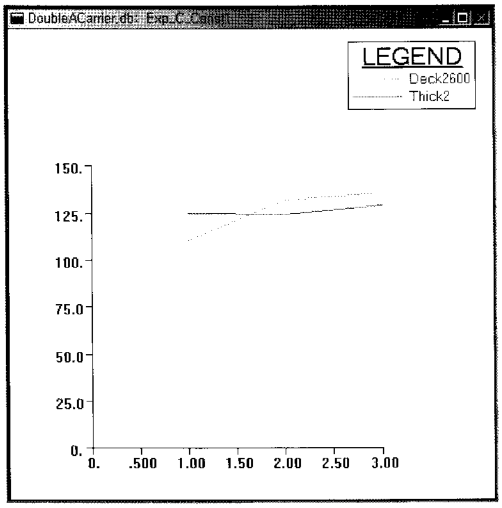


Figure 8: A GUI example to show sensitivity graph.

5 Full container ship example

In this section, the proposed system is applied to a parametric study for a container ship. One of big issues of a large container ship is the use of very thick plates on hatch coaming top. The thickness is strongly dependent on the stress on the plates.

In this illustrating example, which parameter is the most affective to the stress and their impacts on the weight are to be checked together. The conceptual definition of the problem is depicted in Figure 8. Three parameters and two responses are selected for this parametric study.

Parameters
- Thickness of hatch coaming top of three levels, 70t, 74t, and 78t
- Wing breadth of three levels, 2400mm, 2600mm, and 2800mm
- Double bottom height of three levels, 1900mm, 2100mm, and 2300mm

Responses
- Maximum von-Mises stress on hatch coaming top
- Total weight

Thickness of hatch coaming top is a property parameter and the other two parameters are shape parameters. Total 16 experiments are generated automatically using central composite design method. The procedure to define this experiment is depicted in Figure 9. Figure 10 displays 16 experiment results in a graph where two axes represent two responses, respectively. The dotted line can be seen as an approximate Pareto front line. Since four points composing the Pareto front line are superior to other design points, it is reasonable to select one point among them considering the weight budget and available plate thickness.

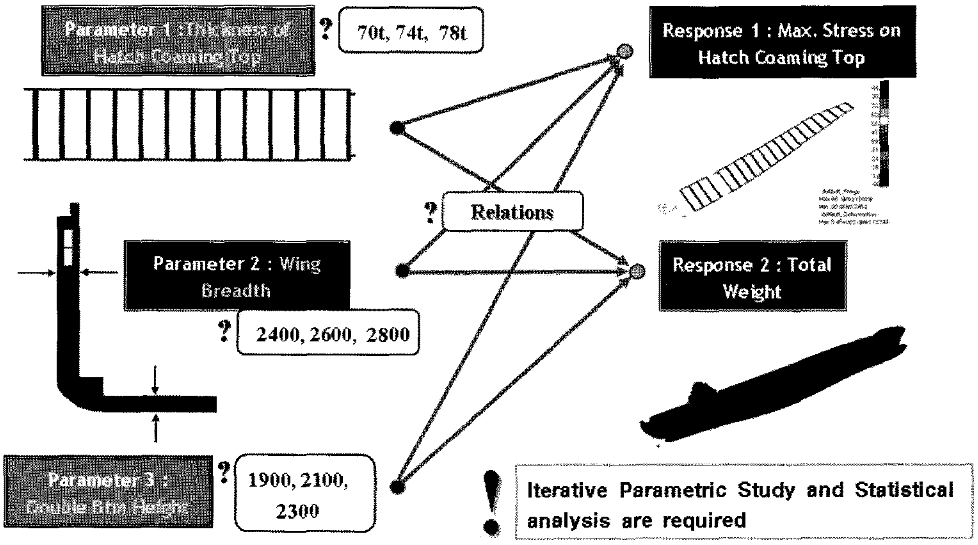
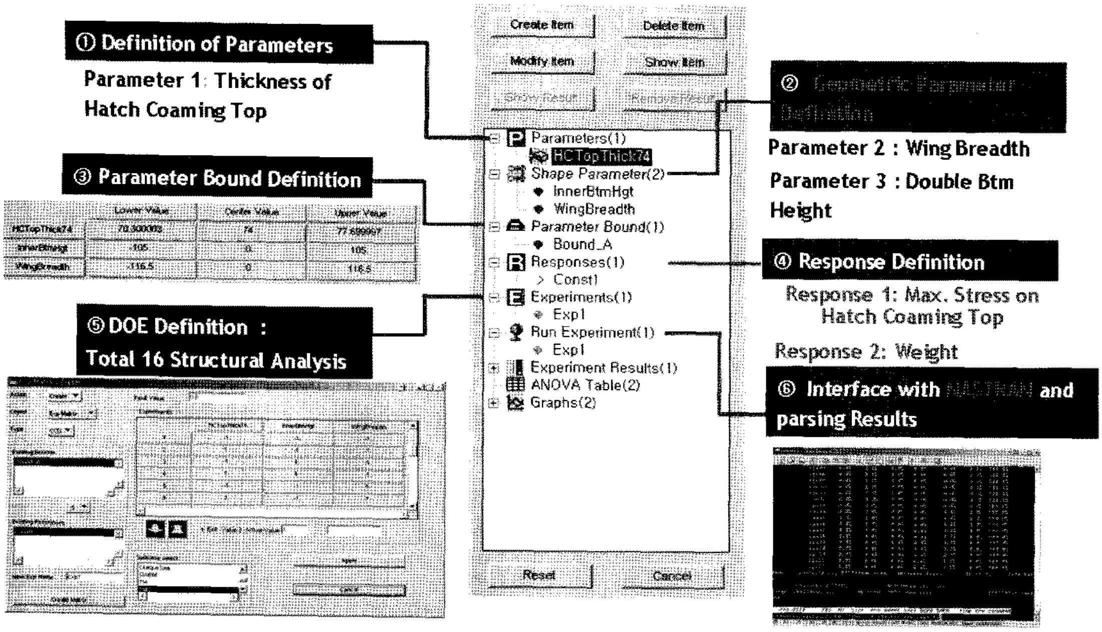


Figure 8: Problem definition of parametric study of a large container ship.



	Lower Value	Center Value	Upper Value
HCTopThick74	70.300003	74	77.699997
InnerBtmHgt	-105	0	105
WingBreadth	-116.5	0	116.5

Figure 9: Parametric study definition processes.

Statistical analysis is performed for the experiment results and it is summarized in ANOVA table. Based on the ANOVA table, two interaction graphs can be plotted to represent the influence level of three parameters on two responses. Wing breadth has the largest impact on the stress on hatch coaming top followed by thickness of hatch coaming top and double bottom height. The effects of three parameters on total weight are similar as shown in Figure 11. Conclusively, it is decided to be the best alternative to increase the wing breadth in order to reduce the stress level on hatch top most efficiently among the three parameters.

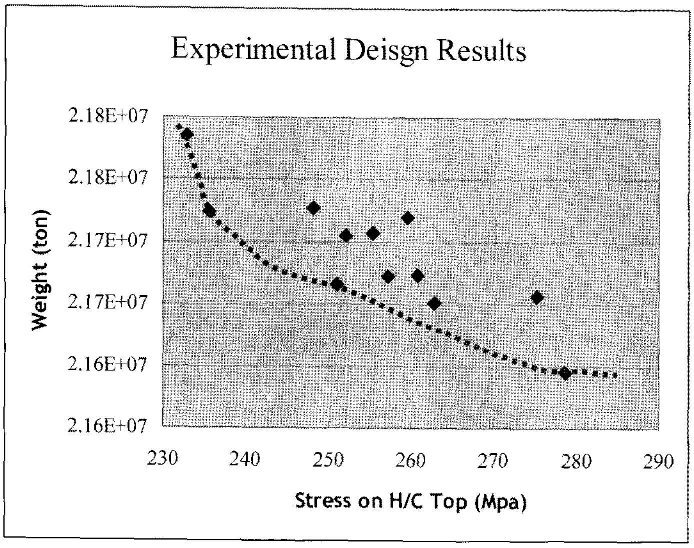
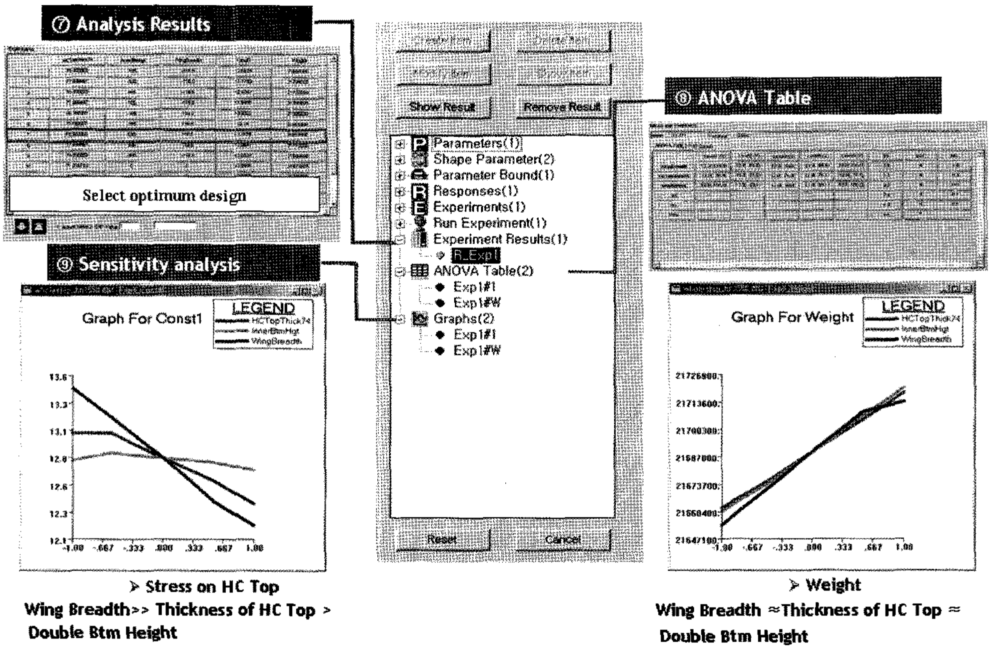


Figure 10: A graph for experiment results.



➤ Stress on HC Top
Wing Breadth>> Thickness of HC Top > Double Btm Height

➤ Weight
Wing Breadth ≈Thickness of HC Top ≈ Double Btm Height

Figure 11: Statistical analysis for parametric study results.

6 Conclusion

This paper proposes a system to facilitate a parametric study using design of experiments (DOE). It is designed to operate on a FE pre-processor (MSC.PATRAN) environment and assists a user to define a parametric study by directly interacting with FE model. Scantlings such as plate thickness or stiffener section profile can be selected as parameters in the parametric study. Simple shape change is also enabled by translating predefined nodes in parallel along x, y or z axis. Any kinds of FE results for any part can be chosen as a response to be investigated. All definitions are made through GUI of FE pre-processor. In addition, since FE model is directly controlled instead of CAD geometry, a burden of mesh generation for strength assessment can be avoided.

Once an experiment is designed, a series of FE analyses are automatically carried out and the experiment results are summarized in a table. An ANOVA table and interaction plots are provided and the relations between parameters and responses can be easily identified.

From a system point of view, all data related with a parametric study can be stored in FE model database and any additional file is not generated. Overall procedures can be easily recognized through a tree type GUI.

Using the proposed system, a user can be assisted to analyze a design problem in easy way and determine design variables considering performances of interest.

References

Engineous Software, Inc. 1999. iSIGHT Designer's Guide : version 5.0, North Carolina, 1999.

Goldberg, D.E. 1989. Genetic Algorithms in Search, Optimization and Machine Learning, Addison Wesley, Massachusetts.

Haftka, R.T., R.V. Grandhi. 1986. Structural shape optimization—a survey, Computer Methods in Applied Mechanics and Engineering. **57**, 91-106.

Huh, J.S. 2000. Development of a Mechanical Design Tool for MEMS Integrating Commercial Code, Master's thesis of KAIST.

Ku, J.K., S.S. Rao and L. Chen. 1998. Taguchi-aided Search Method for Design Optimization of Engineering Systems, Engineering Optimization, **30**, 1-23.

Kwak, B.M. 1994. A review on shape optimal design and sensitivity analysis, JSCE Journal Structure Engineering/Earthquake Engineering, **10**, **4**, 159-174.

Park, S.H. and B.S. Lee. 1995. Developing a Maneuvering Design Capability for Naval Vessels. Proc. PRADS '95, Seoul, pp. 1.616-1.628 Park, G.J. and Arora, K.S. 1987 Role of Database Management in Design Optimization System, Journal of Aircraft, **24**, **11**, 745-750.

Peace, G.S. 1995. Taguchi Methods: A hands-On Approach to Quality Engineering Addison Wesley, Massachusetts.

Phadke, M.S. 1989. Quality Engineering Using Robust Design, Prentice Hall, New Jersey.

Rajeev, S., and C.S. Krishnamoorthy. 1992. Discrete Optimization of Structures Using Genetic Algorithms, Journal of Structural Engineering, ASCE, **118**, 1233-1250.

Salajegheh, E. and G.N. Vanderplassts. 1993. Optimum Design of Structures with Discrete Size and Shape Variables, Structural Optimization, **6**, **2**, 79-85.

Shin, J.H., B.M. Kwak. 2001. Optimization of Machine Tools Structure using a CAD-based Optimal Design System, Proceedings of the KSME 2001 Spring Annual Meeting, 926-931.

Vanderplaats Research & Development, Inc. 1998. VisualDOC Manual : version 1.0, Colorado Springs, CO.

Wu, S.J. and P.T. Chow. 1995. Genetic Algorithms for Nonlinear Mixed Discrete-Integer Optimization problems via Meta-Generic Parameter Optimization, Engineering Optimization, **24**, 137-159.

Yu, Y.G. 2003. Development of a CAD based General Purpose Optimal Design and Its Application to Structural Shape for Fatigue Life, Master's thesis of KAIST.