Transactions on the Built Environment vol 42, © 1999 WIT Press, www.witpress.com, ISSN 1743-3509

# Effective pre- and post- processors for FEM strength analysis using AutoCAD possibilities

M. Bogdaniuk[(1),(2)], A. Kniat [(2)], W. Puch[(2)]
(1) *Polish Register of Shipping, Gdańsk, Poland*
*Email: m.bogdaniuk@prs.gda.pl*
(2) *Technical University of Gdańsk, Faculty of Ocean Engineering and Ship Technology, Gdańsk, Poland*
*Emails: puch@pg.gda.pl, olek@pg.gda.pl*

## Abstract

An advanced software package to calculate and analyse ship hull strength by Finite Element Method (FEM) is being developed in Polski Rejestr Statków in collaboration with the Technical University of Gdańsk – Faculty of Ocean Engineering and Ship Technology and design offices of the main Polish shipyards.
The software incorporates topology, loads, boundary conditions and material properties into the FEM model (a pre-processor facility) and enables to perform detailed analysis of the obtained results ( a post-processor facility).
The system also includes translation modules that allow the conversion of input data into formats required by programs such as SAP, NASTRAN and ANSYS to ensure that the system is compatible with PRS clients existing programs.
The basic advantage of the pre-processor and post-processor is that they are an amendment to AutoCAD computer program. AutoCAD is very popular in Polish shipbuilding industry and it will be very easy for ship designers to learn how to operate the FEM software package.
In the paper the present form of the software package, its improvements and amendments which are being developed to make it fully professional, are described.

# 1 Introduction

In 1996 Polski Rejestr Statków (PRS) took the initiative to create a new software for stress analysis of ship's hull. To achieve the goal, the cooperation agreement between PRS, Gdańsk Shipyard, Gdynia Shipyard, Szczecin Shipyard and Technical University of Gdańsk was signed. All institutions involved could contribute to the initiative and benefit from it. Since all the institutions are deeply involved in shipbuilding a good level of customisation to the industry requirements could be expected. Another advantage of such initiative is that it offers the possibility to fully control the software and easily implement changes in the future. Furthermore students of Technical University of Gdańsk, prior to employment in shipyards, will be familiar with the software. The same software will serve designers at shipyards and surveyors in the classification society. This means that approval of technical documentation by PRS will be more effective.

It was assumed that the software will be applied to strength analysis in elastic range of thin-walled structures. Ship hulls belong to this category of structures. For such scope of application using plate, membrane and bar finite elements is sufficient and only these elements are available in the software.

Preparation of input data for FEM analysis is usually a time consuming task. It is also important to produce understandable reports containing results of FEM calculations. So, it was decided to create only effective pre- and post- processor for the analysis and to use reliable and popular existing solvers of such computer programs as NASTRAN, ANSYS and SAP4. This means that the software being created is able to prepare data defining FEM model of the structure in the formats required by the a.m. programs and to read and make use of results of the calculations made applying these programs. Creating a complete program comprising not only pre- and post- processor but also the solver would be time and cost consuming task. In addition, such programs as NASTRAN or ANSYS are necessary for non-linear calculations which sometimes must be done at design offices of shipyards and at the classification society.

The assumption made at the very beginning was that the software should be easy to operate and effective enough to create a coarse FEM model of entire ship hull structure or fine FEM models of ship structures details. AutoCAD program was selected as a tool which, after special modification, would fulfil the a.m. requirements.

AutoCAD is very popular at design offices of Polish shipyards. This means that designers will be able to easily learn how to operate the pre- and post-processors being created. AutoCAD has also a good software interface for customisation.

It was assumed that complete FEM model of hull structure will be created with standard AutoCAD elements. The material properties, constraints and loads are added to the model with the created software and the software will visualise the results of solver's calculations.

The next problem to solve was to decide about hardware platform and operating system. The following criteria were considered:

- availability;

Transactions on the Built Environment vol 42, © 1999 WIT Press, www.witpress.com, ISSN 1743-3509



- price;
- accessible software interface for customisation;
- efficiency (size of allowable models);
- further development perspectives.

The most advanced professional CAD systems and pre & post processors for FEA work on RISC UNIX workstations. However if RISC UNIX workstations are present in design offices they are usually applied to very specific tasks and are occupied. In addition they are of different make, run different operating systems and different CAD programs. In such case, trying to find some common platform is rather difficult.

Intel PC computers are much more popular. Besides there is usually a big group of users who already know the PC software which shortens the process of training. The final decision was a compromise. As the hardware platform the Intel PC with Windows 95/NT was chosen.

The structure and possibilities of currently available version of the software are described in Ch. 2. In Ch. 3, the efforts undertaken to improve the software and expand its possibilities are mentioned.

## 2 Currently available version of the software

The software operates under DOS using AutoCAD v.12. It allows to process FEM models containing up to 50 000 finite elements.

C and ADS API were used to create the software, basing on AutoCAD. WATCOM C was used as compiler. Such a method enabled to create a reliable software with relatively short time of the execution.

The software covers:

- assignment of material properties to the model;
- assignment of constraints and loads;
- export of data to the solver;
- import of data from solver after calculations;
- deformation of the original model according to calculations results;
- visualisation of stress distribution as colour map or vectors.

Plate and membrane finite elements are created using AutoCAD's option 3DFACE and bar elements using the LINE option. The nodes are not primary members during FEM model preparation. They are generated automatically when the model is exported to the solver. However, for diagnostic purposes it is still possible to check the location of solver's node of given label in AutoCAD model. In this way the user may modify the model in a more natural way and is not expected to keep track of the nodes.

One of the main advantages of the software is an easy way of managing the loads applied to the FEM model. All load values can be set in the form of algebraic expressions, including X, Y and Z coordinates. If expressions are appropriate no changes in load conditions are reqired when modifying the model. Recalculation of expressions after modification assures proper values.

Transactions on the Built Environment vol 42, © 1999 WIT Press, www.witpress.com, ISSN 1743-3509



Expressions can be of quite complicated form what makes the mechanism even more flexible.

The program provides the load condition manager. The manager enables storing, changing, copying and combining different sets of loads. The same model may be easily associated with a chosen load condition and exported to the solver. Each set in the condition can be modified with numeric coefficient and chosen directions can be masked selectively. The role of the load condition manager is to enable the user to easily apply different loads to the same model to carry out calculations and to confirm model's correctness.

The boundary conditions (the constraints) can be managed similarly. In the software there are also many small but useful routines like changing orientation of group of elements or finding elements of given properties. Naturally, all capabilities of standard AutoCAD are at the user's disposal. Different views and viewports, zooming, hiding lines, rendering, coordinate system, point identification and many other tools from AutoCAD can be used to achieve better results in modelling.

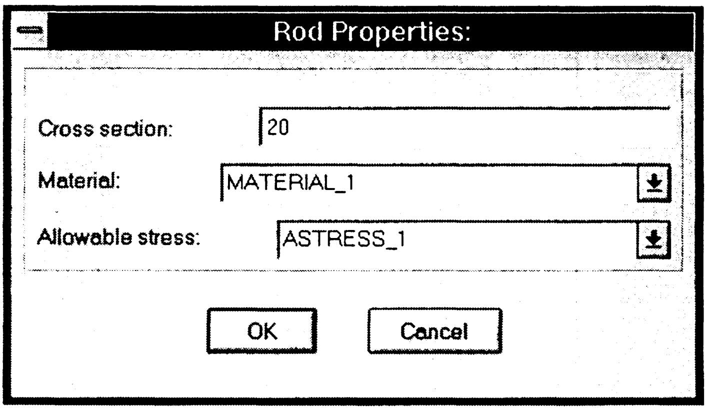
Below the commands accessible in pre-processor, in addition to the AutoCAD interface, are described.

*Material database commands:*

***MATERIALS*** – manages material list;
***ASTRESS*** – manages allowable stress list.

*Creation and modification commands:*

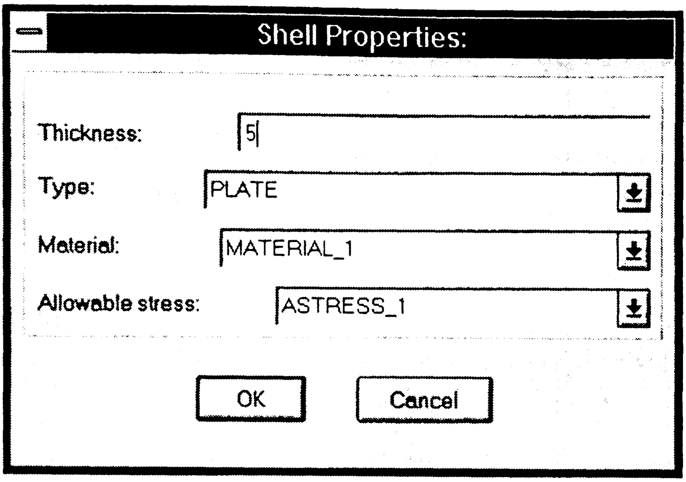
***RODS*** – creates new rods. The dialog box used, typical of the software, is shown here:




***SHELLIS*** – creates new shell. The dialog box used is shown here:

Transactions on the Built Environment vol 42, © 1999 WIT Press, www.witpress.com, ISSN 1743-3509






***ROD_PROP*** – modifies rod's properties;
***SHELL_PROP*** – modifies shell's properties.

*Loads and boundary conditions commands:*

***FORCE*** – creates a new set of concentrated forces and moments;
***PRESSURE*** – creates a new set of pressures;
***CLOAD*** – creates a new set of continuous loads;
***STRESS*** – creates a new set of stress;
***DISPLACEMENT*** – creates a new set of displacements;
***CONSTRAINT*** – creates a new set of constraints;
***SPRING*** – creates a new set of springs.

Each of the above commands requires:

- name of set;
- constant values or formulas which can be created using simple commands such adding and multiplying and the following functions: *R2D(), D2R(), SIN(), TAN(), ASIN(), ACOS(), ATAN(), SINH(), COSH(), TANH(), ASINH(), ACOSH(), ATANH(), LN(), LOG(), LOG2(), EXP(), EXP10(), SQRT(), ABS(), ROUND(), INT()* of *X, Y, Z* variables representing coordinates of nodes;
- set of AutoCAD objects on which loads or boundary conditions are placed;
- cubic space limits within which set values are applied.

*Conditions and sets management commands:*

***CONDITIONS*** – manages load conditions for the current model;
***LOADSETS*** – manages load sets;
***DISPLACEMENTSET*** – manages displacement sets;
***CONSTRAINTSET*** – manages constraint sets;
***SPRINGSETS*** – manages spring sets.

*Export commands:*

*TESTS* – test integrity of the prepared model;

*EXPORT2SOLV* – exports the model prepared in AutoCAD and the selected load condition to the solver.

*Utility commands:*

*SHOW_AXES* – displays and changes local coordinate system of selected objects;

*EXPOSE* – defines a set of objects to be visible, all other objects are hidden;

*SHRINK* – shrinks selected objects. All edges become visible;

*UNSHRINK* – undoes results of shrinking;

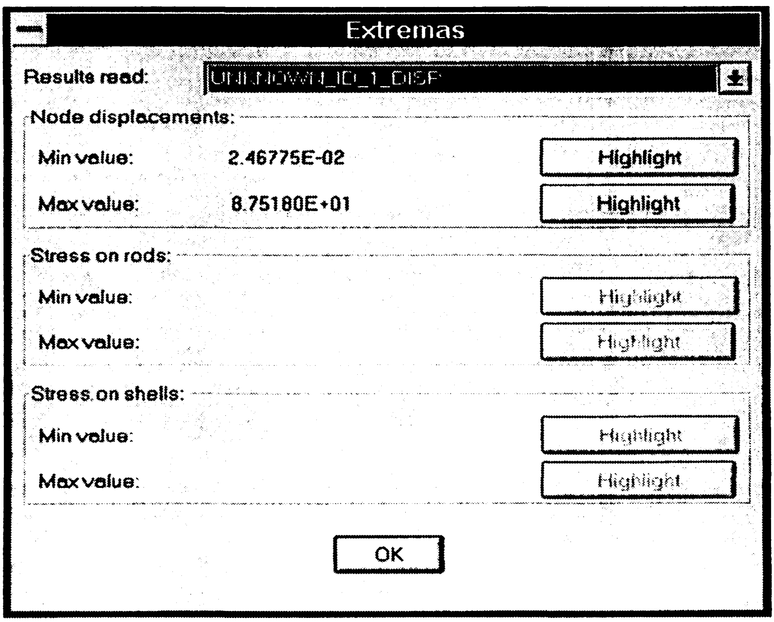
*READFRAME* – reads coordinates of a frame from a text file.

The post-processor commands listed below should also help the reader to imagine possibilities of the system:

*SHOW_RESULTS* – displays results of solver calculations on the whole model;

*SELECT_RESULTS* – displays results of solver calculations on selected objects;

*EXTREMAS* – finds minimum and maximum stress values. The dialog box used, typical of the post-processor, is shown here:



Transactions on the Built Environment vol 42, © 1999 WIT Press, www.witpress.com, ISSN 1743-3509



***RES_DISPLACEMENT*** – creates a set of displacements from the result of solver calculations;
***SELECT_LABEL*** – finds shell, rod or node of given label;
***LIST_STRESS*** – displays numeric stress values;
***MAPS*** – manages maps of colours;
***DRAW_MAP*** – draws a map of colours with corresponding values.

The software was tested by the shipyards participating in the task. Pre-processor results for a flat frame, results of applying SAP4 as the solver and post-processor results were compared with the solution of the same FEM model using such computer programs as ANSYS, NASTRAN, GL FRAME, GIFTS and NISA-II. The results were very similar.

In Fig. 1 the FEM model of a bulk carrier hull structure module is shown. The model was prepared by a group of students as a part of their diploma project. This example proves possibility of applying the software to effective solving practical problems encountered by shipyards in the designing process.

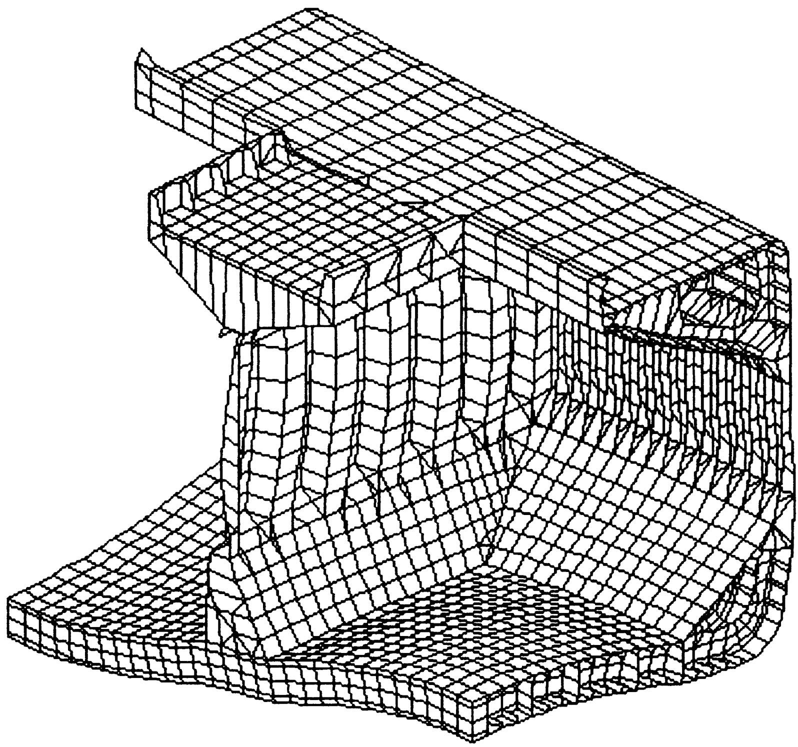


Fig. 1 Deformed FEM model of a bulk carrier module

# 3 Further development of the software

The system is being improved now. First of all the software is beeing translated to operate in WINDOWS 95 NT system and AutoCAD14.

New meshing possibilities will be added. In the beginning the user will create a coarse mesh. Then the initial finite elements will be divided into smaller ones automatically, according to instructions written by the user. This means that creating topology of the model, which usually is the most consuming part of the data preparation process, will be much more effective and quite an easy task.

The software will also be able to perform top-down analysis. First a coarse FEM model of a structure will be solved. Then, if necessary, it will be possible to determine stress distribution in zones of special interest (stress concentration areas, for example). The border of such a zone will be selected by the user. The software will then create a FEM model of the selected zone. The loads, constraints, material properties, etc., will be taken from the coarse model. Displacements calculated for the coarse model will be automatically applied as the boundary conditions on the border of the selected zone. Finer mesh of the selected zone can be generated. In such a situation the boundary conditions and the loads will be automatically corrected to suit the new mesh.

Another advantage of the modified software will be effective tools for creating reports from the FEM calculations. For example, a zone of interest can be zoomed and the user can make that numerical values of mean stress values in the finite elements are written in the elements areas. Such a possibility will be very useful for the designer and the classification society surveyors.

It is expected the a.m. possibilities of the software will be available at the beginning of 2000.

Transactions on the Built Environment vol 42, © 1999 WIT Press, www.witpress.com, ISSN 1743-3509



# 4 Final remarks

The software being developed is restricted to FEM strength analysis in linear-elastic range of thin-walled structures. Such a restriction allows to make it very effective for the input data preparation and reporting the calculation results. It does not constrain the most popular analyses performed in the design offices and the classification society. The software is based on well known, to ship designers, AutoCAD program what makes it easy to learn and operate.

The software being developed will be fully controlled by PRS and the shipyards. This means that futher improvements, if necessary, can be easily made.