ANNALS of the ORADEA UNIVERSITY.

Fascicle of Management and Technological Engineering, Volume VII (XVII), 2008

STUDY OF VON MISES TENSION AND DISTORSIONS , AT HYDRAULIC MACHINERY WITH CLOSED CHASSY

Gh Girniceanu [1], Alin Stancioiu[2], Ghimis Stefan[3]
University "Constantin Brâncuşi", Engineering Faculty,
e-mail:girniceanu@utgjiu.ro

Keywords: Von Mises stresses, dynamic, finit element, displacements

Abstract: The determination of static and dynamic components from a structure, in the context of actual economy, the design of structures must be optim concerning the material and higher productivity. The method used for reducing the time and material consumption is the finite element method which may be introduced and explained from two interlacing viewpoints: one stressing its physical significance, the other its mathematical context.

1. Introduction

The determination of static and dynamic components from a structure, in the context of actual economy, the design of structures must be optim concerning the material and higher productivity.

The finite element method which may be introduced and explained from two interlacing viewpoints: one stressing its physical significance, the other its mathematical context.

The basic concept in the physical interpretation is the breakdown (disassembly, tearing, partition, separation, decomposition) of a complex mechanical system into simpler, disjoint components called finite elements or elements. These degrees of freedom are represented as the values of the unknown functions as a set of node points. The element response is defined by an algebric equations constructed from physical arguments. The response of the original system is considered to be aproximated by that of the discrete model constructed by connecting or assembling the collection of all elements.

If the behavior of a system is too complex, the recipe is to divide it into more manageable subsystems. If these subsystems are still too complex the subdivision process is continued until the behavior of each subsystem is symple enough. These "primitive pieces" are called elements, which will interact giving the behavior of the total system.

In the mathematical interpretation, the FEM is viewed as a procedure for obtaining numerical approximations to the solution of boundary value problems for a domain Ω. This domain is replaced by the union U of disjoint subdomains $\Omega^{(e)}$ called finite elements. In general the geometry of Ω is only approximated by that of $U\Omega^{(e)}$.

2. Steps in finite element analys

Using a specialized software for Finite Element Analys like Cosmos/M, Ansys, Nisa the structures can be analyzed. The steps involved are: preproccessing and postproccessing.

Preproccessing is the initial step when the geometry is created, the boundary conditions are applied and the desired type of analys is choosed using a solver.

Postproccessing is the step when the results are viewd as 2D or 3D graphs or color maps .

A important step in obtaining good results (after building geometry) is meshing the structure when it's divided in elements which aproximate the structure shape depending on number of elements used for meshing the structure, figure1.

If meshing errors appear like in figure 2, the mesh refin can be increased using more elements or curve elements figure3.

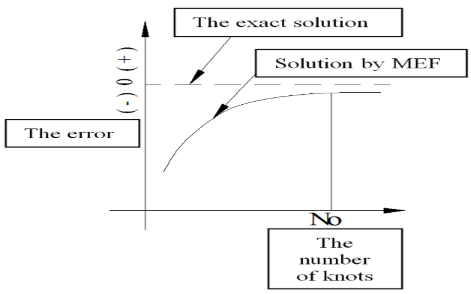

Figure 1. The selection of number of nodes

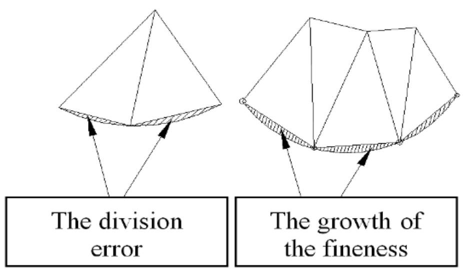

Figure 2. Digitization error

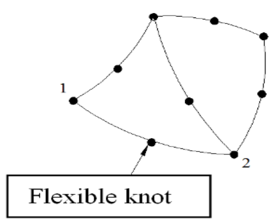

Figure 3.The elements or curve

The most used elements is structure mechanics are: unidimensional, bidimensional, tridimensional elements.

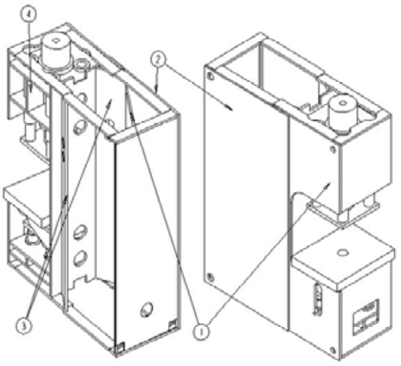

Figure 4. The structural scheme of the hydraulic press

If the structure is too complex as geometry (entities are hidden and the mesh can't be viewed, figure 4), the mesh quality can be evaluated through parameters.

After the mesh is created and displayed, options are available for displaying any areas on the mesh that are imperfect. The following parameters can be used to evaluate the quality of mesh:

- High aspect ratio-Occurs with long thin elements. An aspect ratio is a measure of a mesh element's deviation from having all sides of equal length.

- Entering an overly large value for the Global Minimum may cause tet mesh elements to be created with high aspect ratios.

- Low Jacobian distortion index-Occurs when 10-node tetrahedral elements have too much curvature. A Jacobian distortion index is a measure of how distorted an element is.

- High mid-ratio-Occurs when 10-node tetrahedral elements have too much curvature. The mid-ratio is Curvature divided by ``Length'', where:

Curvature is the distance from the mid-node to the straight line connecting two corner nodes

Length is the distance between the corner nodes.

The mid-ratio value is used primarily by the ANSYS FEA solver.

The Aspect Ratio R for both tetrahedron and quadrilateral elements is calculated according to the following formula:

$$R = E/h$$

where E is the longest edge and h is the shortest height (the distance between a vertex and the opposite surface or edge.)

The Distortion is defined as the ratio between the maximal Jacobian and the minimal Jacobian calculated at the four Gaussian integration points. This test is performed on parabolic elements only.

The Mid Ratio M is defined according to the following formula:

$$M = h/L$$

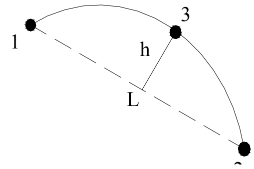

Figure 5. Parabolic connection

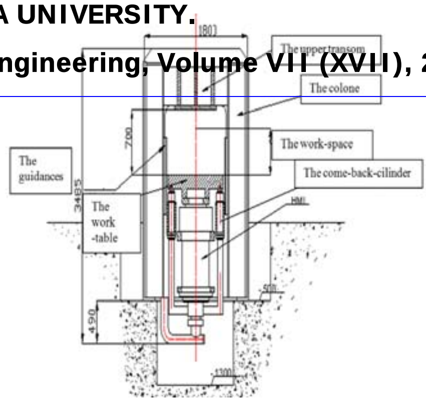

Figure 6.Functional scheme of the hydraulic press

where L is the distance between two adjacent corner nodes and h is the distance between the mid-node and the connecting line of the corner nodes. This test is performed on parabolic mesh only figure 3,5.

These parameters are important because ignoring them cause the results to be incorrect.As example is considered the structure from figure 6.

A displacement analys will be performed for the lateral walls and superior beam because a big displacement of these influence the process of metal forming. The units system used is millimeter Newton Second (mmNs).

The material used is steel and its characteristics for the static analys are:

$E_x = 2{,}1 \cdot 10^5$ [N/mm^2] ; $\nu_{xy} = 0{,}28$; $\rho = 7{,}829 \cdot 10^{-9}$ [t/mm^3] and g =9806,65 [mm/s^2]

First, the geometry was created using a 3D modelling system (Pro/ENGINEER 2000i) see figure 7, where 1 superior beam, 2 lateral walls, 3 shell for fixing the principal cylinder.

Using Ansys 5.5.1 the structure is meshed using tethraedal elements because these elements take stresses and generate less mesh errors

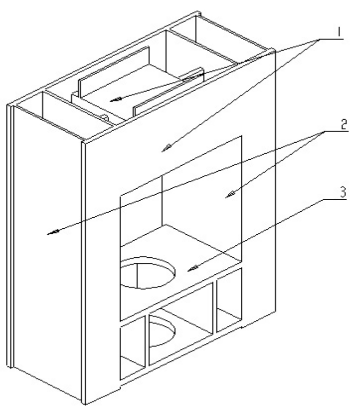

Figure 7.Geometrical elements of the body

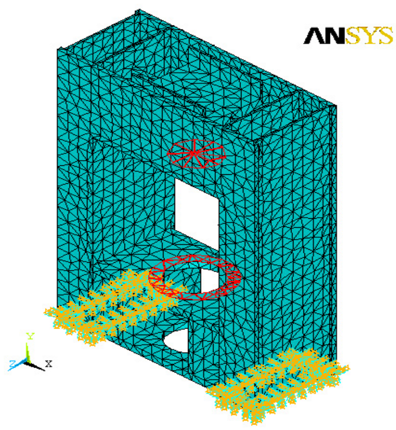

Figure 8 Frame digitization

3. Conclusions

In figure 8 is shown the mesh geometry where it can be observed the boundary conditions too, these ending the preproccesing step. Using the right solver it can be performed a fast finit element (FFE) analys which shorten the time needed for solving. The next step is postproccesing where the displacements after Ox (figure 9, 10) and Oy (figure 11) axes, and Von Mises stresses (figure 12,13 and 14). The determination of static and dynamic components from a structure, in the context of actual economy, the design of structures must be optim concerning the material and higher productivity.

The method used for reducing the time and material consumption is the finite element method which may be introduced and explained from two interlacing viewpoints: one stressing its physical significance, the other its mathematical context.

Through the imposed conditions from the project theme profile tolerance (edge, surface), static and dynamic rigidity conditions and from the analys results it can be draw the conclusion that the structure is rigid enough.

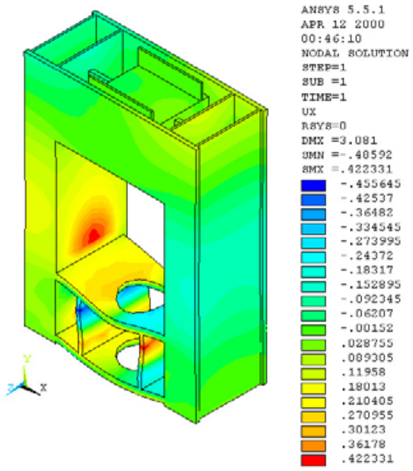

Figure 9 Ox distorsion

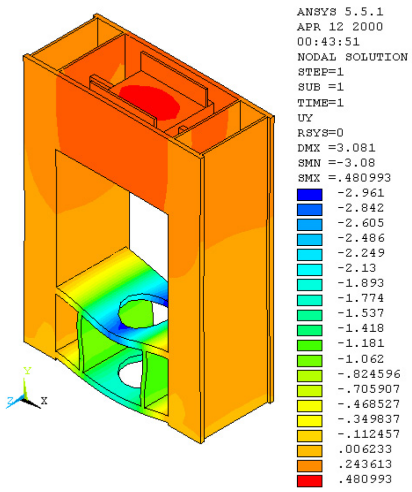

Figure 11 Ox distorsion

Figure 10 Ox distorsion

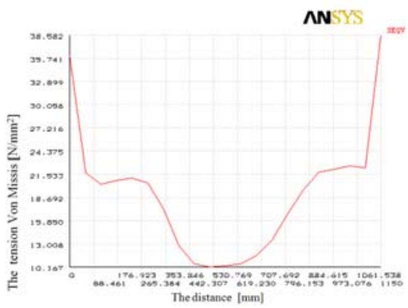

Figure 12 Von Misses pressure

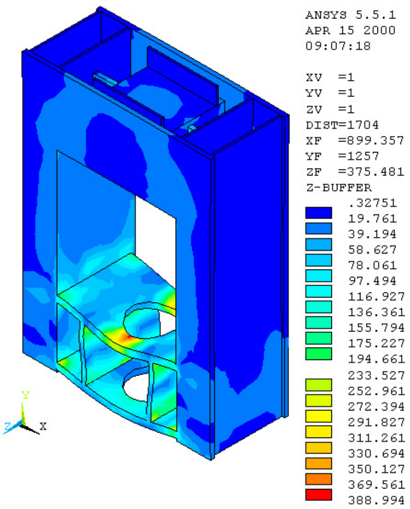

Figure 13
Von Misses pressure

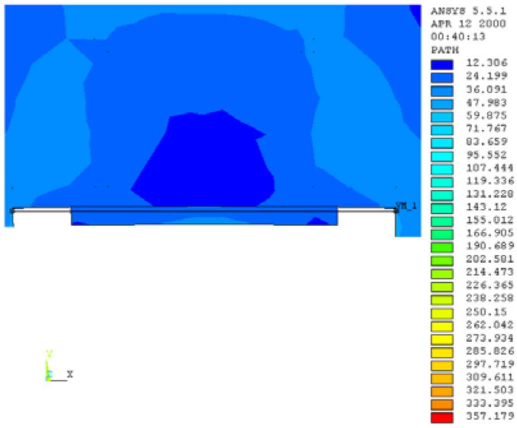

Figure 14.
Von Misses pressure

4. Bibliography

[1] Robert D. Cook - Concepts and Application of finte Element Analysis, John Wiley & Sons, New York, 1981

[2] C. Pacoste ş.a - Metode moderne în mecanica structurilor, Ed. Ştiinţifică şi Enciclopedică, Bucureşti, 1988

[3] N. Posea - Calculul dinamic al structurilor, Ed. Tehnică, Bucureşti, 1991

[4] * * * ANSYS Online Manual Release 5.5

[5] * * * Pro/FEM- POST (TM) User Guide