Nedelcu Dorian
Associate Professor

Mănescu Ştefan Tiberiu
Professor

Câmpian Constantin Viorel
Professor

The „Eftimie Murgu" University
Resita, Faculty of Engineering
Romania

Finite Element Analysis Through COSMOS M/Design STAR

The paper present one of the most performing program that uses for analysis the finite element method (FEM) Cosmos/M Design Star, produced by Structural Research and Analysis Corp. The aim of the study is to present the general characteristic of the program, as well as its analysis possibilities.

Keywords: Analysis, Finite element, CAD, Stress

1. INTRODUCTION

Projecting a part must answer to some imposed proportional and functional criterions. The dimensional aspect of projecting is much easier to square up referencing to the functional one, because it represents an engineering conception put across by shape, dimensions, and content. In the wake of it results the equivalent of the real part put across by an execution drawing, allowed to be the universal communication language of engineers. The classical dimensioning theories involve model scheduling and application of some simplified hypothesis, imposed by the applied calculus method. The possibility of experiment on models can offer an answer much closer to reality, but with the price of a large time consume, manual labor and if course, additional costs, fact that it isn't always profitable.

Certainly, a saving alternative to these problems is the simulating and functional analysis programs studied on different fields. These kind of programs offer to projector powerful instruments of study and analysis, which allow technical substantiation to the optimal solution by selecting this one from the numerous variants that can be generated by the computer.

We propose in the following lines to make a descriptive presentation of the possibilities offered by the program, with a synthetic instance of a linear static analysis applied to a card punch handle, moderated with Microstation Modeler (Fig.1)

2. AVAILABLE ANALYSIS TYPES IN COSMOS M / DESIGN STAR

COSMOS M / Design STAR is an performance analysis program with finite parts, in which are available the next analysis types:
- Linear and nonlinear static analysis;
- Modal analysis: own frequency and pulsation modes;

- Lateral flexion calculus;
- Linear and nonlinear warmth transfer in permanent and transitory regime;
- Fluid flowing;
- Electro-magnetically analysis.

We will relegate to 3.0/2201 version of the program, developed by Structural Research & Analysis Corporation, U. S., a specialized program in analysis with finite parts of the parts and ensemble moderated in system majority CAD existing in present.

3. GENERAL CHARACTERISTIC OF COSMOS M / DESIGN STAR

We will cite the main traits of the program:
- Uses Windows Graphical User interface (GUI);
- Operates with multiple documents simultaneous opened, with multiple windows collocated to the same document with multiple overview in any window;
- Operates with moderated geometry through graphical kernels Parasolid, ACIS and STEP AP203;
- OLE technology (Object Linking & Embedding) integrated with the most popular programmers: Word, Excel and MathCAD;
- Automatic generation of reports in HTML format, which contains all textual, numerical and graphical information associated to the analysis;
- The stresses are applied to the geometry, not to nods or parts;
- It offers the possibility of defining some local reference geometries (points, axes or plans) which can be used for defining stresses and stints;
- A great liberty in units using, having the possibility of defining an implied measuring units system, as through the liberty of individual selection to filling in sizes and displaying results.

The finite parts analysis finally leads to an equation system, to whose size depends of the complexity of analyzed model and to whose solution necessitates the application of some numerical performance methods for result calculus. Design STAR uses particular precise and productive algorithms ("solver"), which works 100 times faster then classical resolving technologies. The program offers the possibility to use 3 methods

"solver": "Direct Spars", "FFE" and "FFEPlus", correlated with the analysis type performed, the last one using advanced techniques of matrix primness, recommended to be used for complex problems.

The hardware need of the program implies: Windows operating system (any version), minimum 64 MB RAM, about 90 MB for setup and around 200 MB available work space.

The working technique used by the program dictates the part or ensemble modulation in a specialized projecting program, model that had to be saved from the CAD program in a compatible format from the available list that Design STAR uses: file COSMOS /Design STAR type "part" (*.DgxPrt) respective "Assembly" (*.DgxAsm), files Autodesk Inventor (*.ipt, *.iam), files Solid Edge (*.par, *.psm), files Solid Works (*.sldprt), files Parasolid (*.x_t,*.x_b, *.xmt_txt,*.xmt_bin), files ACIS (*.sat), files STEP (*.stp, *.step), files IGES (*.igs, *.iges), files Pro/ENGINEER (*.prt), files CATIA (*.mod, *.model).

Geometrical rewrites can't be made with Design STAR, exception makes only suppressing or not some constituent parts of the model, operation that involves taking them or not in analysis calculus. There is also the possibility to make a connection between CAD programs and Design STAR, having as effect self-adjusting of stints, stresses, and others specifications in response to the geometrical modifications made by CAD program. In this version, the connectivity option is available only for Solid Works, Solid Edge, and AutoDesk Inventor.

Design STAR offers the possibility using functional curves, punctually defined, specific to the type of analysis broached: temporal curves for nonlinear and transitory thermal analysis, that can describe the variation in time of stresses and frontier conditions, thermal curves for structural and thermal analysis, which can describe the dependence of material properties in function of temperature and used curves in fluid flowing, that can describe debit in function of pressure for pumps and ventilators.

4. DESIGN STAR INTERFACE

The Design STAR interface contains parts of graphical interface specific to Windows: main window, menu, tools cases that can be anchored on the main window sides, as well as the status bar dedicated to informative and "help" type messages display.

The *"Standard"* case contains specific icons in handling documents: *"New", "Open", "Save", "Delete", "Copy", "Paste", "Print", "About", "Help"*.

The *"View"* case contains specific icons to modify the displaying mode of the respective model of graphical results: visualization through enlargement or diminution of a portion or the whole model, by rectangular zoom, static and dynamic, 3D gyration, view translation, model display in *"wire frame"*, hide lines, *"shade"* by applying light sources, perspective, color change to the parts.

The *"Geometry Tools"* case contains icons for connecting with CAD systems, the update in Design STAR of modified geometry in CAD program,

displaying information about weight proprieties, measurement of geometrical entities, applying informative text notes.
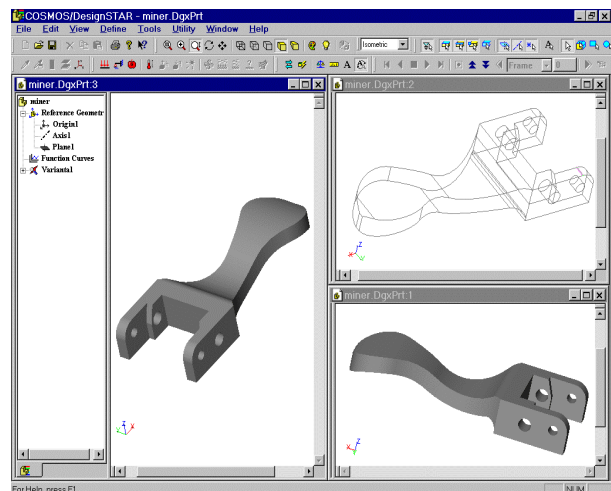

Figure 1.

The *"Selection"* case contains icons for geometrical entities selection, pursuant to the next selection modes: **normal** (which identifies automatically entities frontier by a left mouse click), by **rectangular** or **circular wire frame**, respective *"Tangent Faces and Edges"*, which can select a group of faces or edges being in tangent contact.

The *"Loads/BC"* case contains specific icons for applying stresses and restrains over the model.

The *"Results"* case contains icons for results visualization: display coordinates and values in specific points, display text information and save to files, modifying the color maps, number control and sections orientations through the model and their visualization.

The *"Orientation"* case contains icons for specification of the visualization mode pursuant to standard views: plan – *"Front", "Back", "Top", "Bottom", "Left", "Right"*, 3D – *"Isometric"*, horizontal and vertical mirror.

The *"Animation"* case contains icons for visualization of results animation and saving in AVI format.

The main window is divided in 2 zones:

- *"Visualizer Tree"* zone, containing in tree form the characteristic of the study of case, putted across by representative icons;
- *"Graphic Display"* zone reserved to model handling and graphic display of the analysis results, in color maps shape.

5. DEFINING THE STUDY OF CASE

Design STAR operates with the *"Design Study"* concept, which is technical study of case, understood as a reunion of specific characteristics to an analysis: the analysis type and associated options, materials, a set of stresses and frontier conditions, as well as model disparagement.

First leg, after opening the file that contains geometric modulation, is defining the study of case (Fig. 2), by specifying its name, analysis type (static,

frequency, lateral flexion, thermal, fluid flowing, nonlinear analysis), respective the disparagement mode (with solid parts or *"shell"*, dedicated to surfaces analysis).
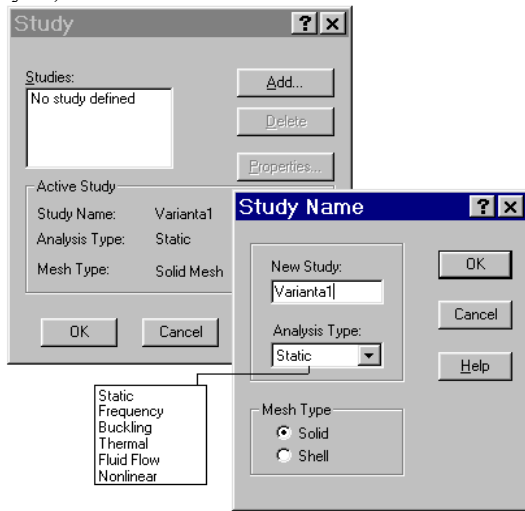

Figure 2.

After defining the name of the study of case, it can be specified its proprieties, which depend of the chosen analysis type (Fig. 3 for static analysis). The main proprieties, correlated with analysis type are shortly described in the following lines:

1) Static analysis

a) Including thermal effect resulted from prescribed temperatures (defined as thermal stress), from the temperature profile generated by an earlier thermal analysis, in permanently or transitive regime or as result of a uniform temperature applied to the whole model.
b) Considering fricative due forces contacts surfaces, calculatingly through the multiplication natural forces generate to the elements contact with the friction coefficient value specifies.
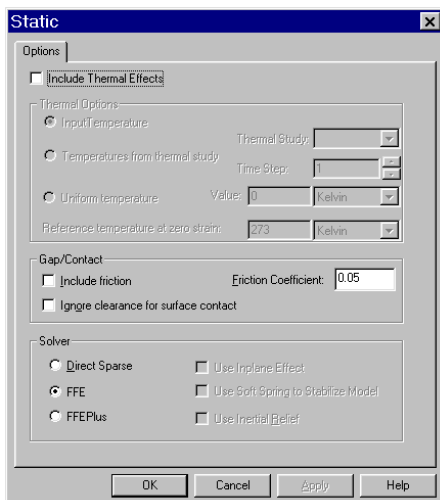

Figure 3.

2) Modal analysis

The numbers of own calculable frequency (implicit 5) and specifying the superior limit of the frequency domain;

3) Buckling calculus

The number of calculable lateral flexions

4) Thermal analysis

Result type: permanent or transient (specifying start and final time and the time increment), the iterative resolving technique: Newton - Raphson scheme, initial temperature and calculus tolerance;

5) For non-linear analysis or fluid flowing are available specialized options.

After specifying the study of case proprieties, *"Visualizer Tree"* zone is completed with *"Components"* icons, (which contains subordinated icons for each part, - *"Part"* to ensemble), *"Loads/BC"* dedicated to loads and frontier conditions, respective *"Mesh"*, disparagement dedicated, all these corresponding to the defined study of case.
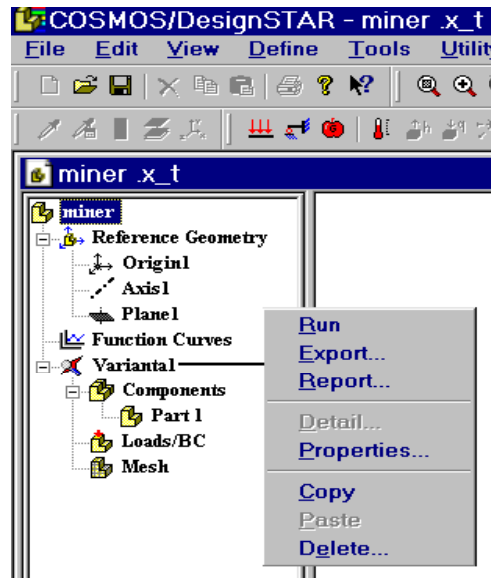

Figure 4.

New study of case can be anytime attached or can be modified the existing ones, for analyzing new variants. Accessing through the right button of the mouse the name of the study of case offers a contextual-sensitive menu (Fig. 4), with specific options handling the studies of case: *"Run"* - the calculus associated with the study of case, *"Export"* - study export to other specialized programs in finite part analysis (GEOSTAR, ANSYS, NASTRAN), *"Report"* - WEB reports generating in HTML format, *"Detail"* - display active options for the selected study, *"Properties"* - visualizing or modifying study properties, *"Copy", "Paste", "Delete"* - study handling by copying, adding in the top of the *"Visualizer Tree"* or deleting.

6. MATERIAL DEFINING

For analysis calculus it impose materials specification for all the parts of the analyzed model, which can be specified by contextual-sensitive menu activated by right-button mouse click: the material is applicable to all components *"Apply Material to All"* - (Fig. 5a), if the domain is activated on the *"Components"* icon, respective to a constituent single

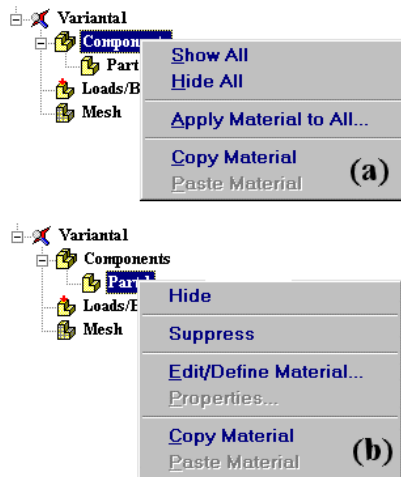part *"Edit/Define Material"* - (Fig. 5b) if the domain is activated on concordant *"Part"*.

Figure 5.

The same menu offers the possibility to hide or re-display (only visualizing, not in disparagement point of view), to one *"Hide/Show"* or all *"Hide all/Show all"*, handling material properties by copying from one study to another *"Copy material/Paste material"*, *"Suppress"* or *"Unsuppress"*, having as effect eliminating or not the part from calculus point of view.

Material can be selected from the bookshop of material provided with the program, which contains materials categorically organized: steel, iron, aluminum, copper, etc.., each category containing a set of material with pre-defined properties. That is why can be selected the material model from the available list: elastic linear isotropic, orthotropic, anisotropic, respective options for nonlinear analysis.

7. DEFINING LOADING AND FRONTIER CONDITIONS

For the analyses calculus is enforced the definiteness of loading and frontier conditions that can be specified through menu options or from the contextual highlighted menu through click on right button mouse to the *"Loads/BC"* from *"Visualizer Tree"* zone, (Fig. 7), which offers the next options: "*Show All*"/ "*Hide All*" – for display or not, apply the frontier conditions, the loading, gravitational forces, temperatures, respective through options "*Restraints ...*", "*Load ...*", "*Body Loads ...*", "*Temperature ...*".
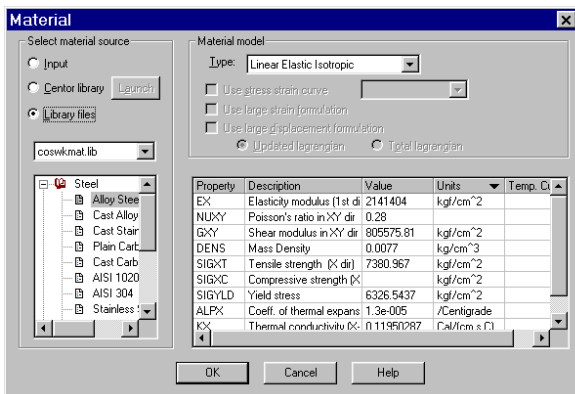

Figure 6.

Are available 4 loading and frontier conditions types:

- Movement restriction for structural studies(static analysis, buckling and frequency); it can be prescribes movements with null values or non-null applicable on vertex, edges or faces;
- Loading for static analysis studies type, buckling and nonlinear, respectively optional for frequency, what includes forces, moments, uniform pressure, gravitational force (linear acceleration, angular speeds and accelerations), thermal loading;
- Loading and frontier condition for studies of thermal type: prescribed temperatures, convection, thermal loading(flux of heat, power or volume) and radiation;
- Loading and frontier condition for analysis of fluid flows: speed, pressures, curve of operation, scalars, temperatures, etc.
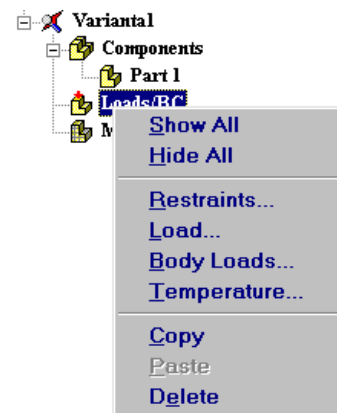

Figure 7.

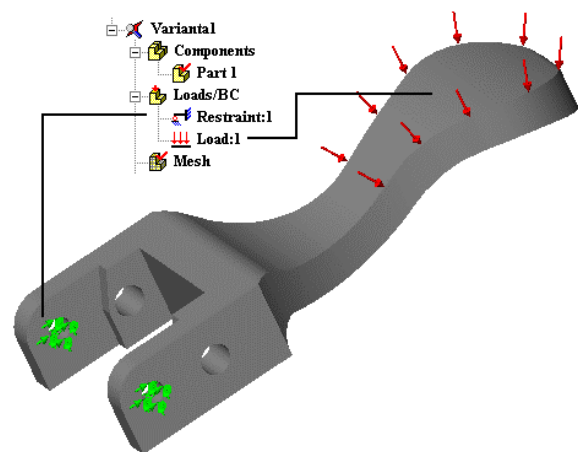For the analyzed model, the loading and the frontier conditions are presented in Fig. 8.

Figure 8.

So, the movement restrictions are prescribed by "*Restraints...*" option, from the sensitive contextual menu (Fig.7), which activates the "*Restraint* " window (Fig.9). The "*Select Geometry*" zone is reserved to selected entities display (type and identifying number) on which will be applied the imposed restriction. The applicable restriction types are pre-defined in the *"Restraint Type"* list:

- "*Fixed*" –spatial fixing selected entity;

- "*Symmetric*" – only for plan faces, forbids plan faces points displacement on normal direction to the plan face, admitting only displacements in the symmetry plan;
- "*Non-Sliding Face*" - only for plan faces, forbids points displacement in the face plan admitting only displacements on the normal direction to the face plan;
- "*Sliding Face*" - only for plan faces , forbids plan faces points displacement on normal direction to the plan face, admitting only displacements on the face plan; (similar with "*Symmetric*" option);
- "*No Translation*" – impose value 0 for X, Y, Z components of the ale displacement;
- "*No Rotation*" – impose value 0 for components X, Y, Z of the rotation;
- "*Prescribed*" - allows prescribed values imposing for rotations and translations, on X, Y, Z directions, rotating option beeing available only for disparagement with "*shell*" type elements; for each type of restriction it can be selected the measure unit form the controls "*Units*".

Each type of applied restriction generates an icon in the study of case's tree (Fig. 8) subordinated "*Loads /BC*" icon, whom right mouse click button accessing offers the option to modify / erase the restriction.

For the analyzed model, was imposed the "*Fixed*" restriction on the cylindrical part of the two holes, graphic represented in Fig. 8.
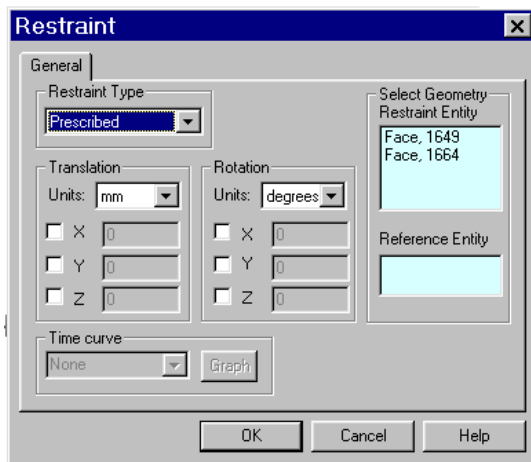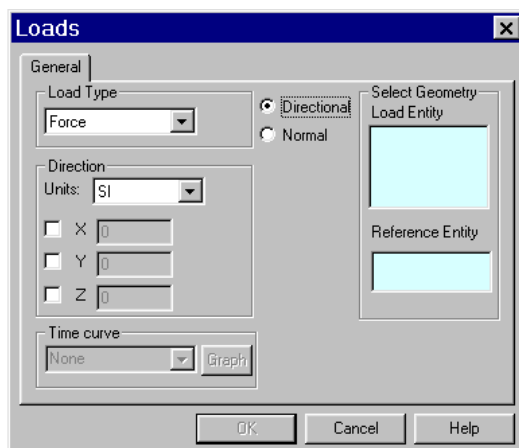

Figure 9.

Figure 10.

The loading can be prescribed by "Load." option, from the sensitive - contextual menu (Fig. 7), which activates "*Loads*" window (Fig. 10). The "*Select Geometry*" zone is reserved to selected entities display (type and identifying number) on which will be applied the imposed loading. The applicable loading types are pre-defined in the "*Load Type*" list:

- "*Force*" – allows applying of a total force on one or several selected entities: vertex, edge or face;
- "*Torque*" – allows applying of a *torsion moment* on one or several selected faces;
- "*Moment*" – allows applying of a moment on one or several selected entities: vertex, edge or face;
- "*Uniform pressure*" – allows applying of a distributed pressure on the selected face;

The applying direction of the loading can be normal or directional, by specifying valor components on the 3 axes. For each loading type it can be selected the measure unit form the controls "*Units*".

Each type of applied loading generates an icon in the study of case's tree (Fig. 8) subordinated "*Loads/BC*" icon, whom right mouse click button accessing offers the option to modify / erase the loading.

For the analyzed model, was imposed the "*Force*" loading on the superior face of the handle, graphic represented in Fig. 8.

Only for nonlinear analysis, "*Time Curve*" control (Fig. 9 and 10) allows to specify a curve of time punctual defined, associated with prescribed values for displacements or loading.

8. THE MODEL DISPARAGEMENT IN FINITE PARTS

The finite parts method impose model devising in a multitude of little parts with simple geometrical forms, defined as **finite parts**, connected in common points called **nods**, so that the whole model is behold, from analysis point of view, as a network of interconnected parts. From analysis point of view, the finite parts method numeric accounts the different behavior of each part depending of the imposed conditions, the global behavior of the model beeing calculated from the totality of calculated information for all the finite parts of the model.

The established term for disparagement is "*mesh*". The disparagement quality is especially important, influencing the final results of the analysis. Design STAR offer the opportunity of automatically generation of disparagement, considering a global ratting value of parts calculated depending on the volume models, them surface and information of geometric type. The disparagement size (number of nods and parts) so depends on model geometry and dimensions, on the selected order for parts ("*Draft*" or "*High*") (Fig. 11), tolerances, the disparagement type and specifications referring to the contact.

Therewith, Design STAR permits local specification of parts sizes by the user, with different value from the global one, in approaching vertexes, edges or faces, for a more precise approximate of the delicate zone, from geometrical point of view "*User-defined Mesh Control*".

In defining the study of case (Fig. 2) are available 2 types of disparagement: *"Solid"* type and *"shell"*.

Disparagement with *"solid"* type parts offers 2 alternatives :

- *"draft quality mesh"* - by generating of *tetraedrical* linear solid element, defined through 4 nods connected through 6 linear edges;
- *"high quality mesh"*- by generating of *tetraedrical* parabolic solid element, defined through 4 nods, 6 semi-nods respective 6 linear edges;
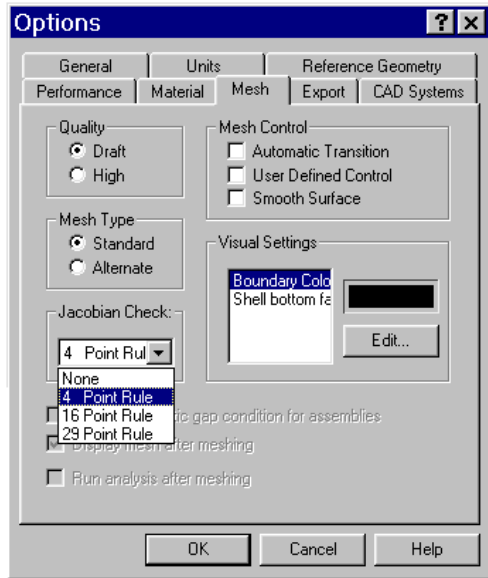

Figure 11.

It is obvious that the best mathematical approximation is offered by the second variant of disparagement, but with the effort of a superior calculus. Each nod has for these *"solid"* part type, 3 degree of freedom, which are the translations along the 3 directions, X, Y and Z, excepting thermal analysis. For thermal analysis the number of degree of freedom is 1, that is temperature.

Disparagement with *"shell"* type parts is applicable to surface models and offers also 2 alternatives :

- *"draft quality mesh"* - by generating of triangular linear element, defined through 3 nods connected through 3 edges;
- *"high quality mesh"*- by generating of triangular parabolic element, defined through 3 nods, 3 semi-nods respective 3 parabolic edges;

The thickness is deduced from the model geometry. Each nod has for these *"shell"* part type, 6 degree of freedom, which are the rotations and the translations on the 3 directions, X, Y and Z, excepting thermal analysis. For thermal analysis the number of degree of freedom is 1, that is temperature.

The *"Mesh Type"* option from the *"Options"* window (Fig. 11), offers the possibility to select the disparage-ment scheme variants: "*Standard*" respective "*Alternate*", the first one beeing much formant and available in this version; the second one is available in the previous versions.

The *"Jacobian Check"* option allows the number of integration points selection (4, 16, and 29) used to verify the parts level distortion.(unavailable for disparagement with *"shell"* parts).

The disparagement control , "*Mesh Control*" zone (Fig. 11) offers 3 possibilities:

- "*Automatic Transition*" – activating this control, the program applies disparagement to details, holes, geometric portions with small dimensions comparative to the whole model;
- "*User Defined Controls*" – activating this control, the program applies disparagement specifications to all zones explicit defined by user (vertex, edges, faces), others than the ones resulted from the global disparagement applied to the rest of the model;
- "*Smooth Surface*" – activating this control, the program refines disparagement , in the sense of reallocation of the frontier nods, to improve the quality of initial disparagement.

The launching of the disparagement process can be made from the main menu of the program, in this order "*Define*" →"*Mesh*"→ "*Create*" or from the sensitive-contextual menu of "*Create*" icon. By *"Apply Control"* options can be specified by the user local characteristics of the disparagement.
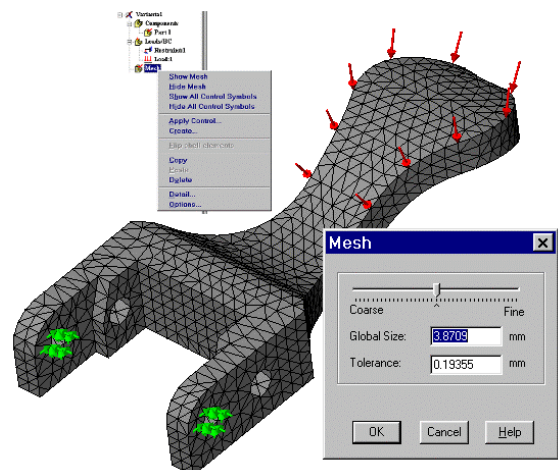

Figure 12.

In many applications, different faces of the parts from an ensemble can come in contact. Principally, after this moment linearity hypothesis (the answer proportionality in report with the loading) it isn't correct because of the new forces generated in the contact zone, the problem becoming nonlinear. Yet, the static analyses can straight contact problems, offering in this way the response prediction of the model.

The contact option is available only for static or thermal analysis and only for *"Solid"* type disparagement of an ensemble, and offers this three possibilities:

- "*Touching Faces: Bonded*" – in which the faces in contact will be coerced in the contact points;
- "*Touching Faces: Free*" – in which the faces in contact will be free to move in any direction; can be obtained theoretical mutual penetration;
- "*Touching Faces: Node to Node*" – in which the program creates interstitial elements , the connection corresponding to the nods from the faces in contact; the faces can displace one in report with the other,

conserving the physical request of avoiding penetration.

9. ANALYSIS CALCULUS

After run over all previous stages presented, is enforced the analysis calculus for obtaining the results, from the main menu, through the option "*Define*" → "*Run*" or "*Define*" → "*Run All Studies*" or from the sensitive-contextual one, accessed with the right button of the mouse on the corresponding icon to the selected name from the analyzed study of case, (Fig. 4). After digital processing of the analysis can be visualized its results. The user hasn't any check during the analysis, excepting windows that indicate under the form of a thermal bar the calculus evolution.

10. RESULTS DISPLAY

The results can be consulted under graphical form (color maps and diagrams) respective numerical values. The results depend on the made analysis type: results regarding the tensions and stresses, displacements, deformations, comparative checking by imposed admissibility criteria, thermal results - temperatures, temperature gradient, thermal flow, etc.

The *"Results Visualization"* case offers the necessary instruments for result display.

The graphical results put across color maps, having color legends associated with digital value domains (Fig.13). The maps can represent colored outlines, vectors that signifies magnitude and the sizes direction, section or zone with constant enforced values.

Digital values can be obtained punctually or for different entities(edges, faces, parts, etc.). Digital result can be visualized tabular and / or saved in files.
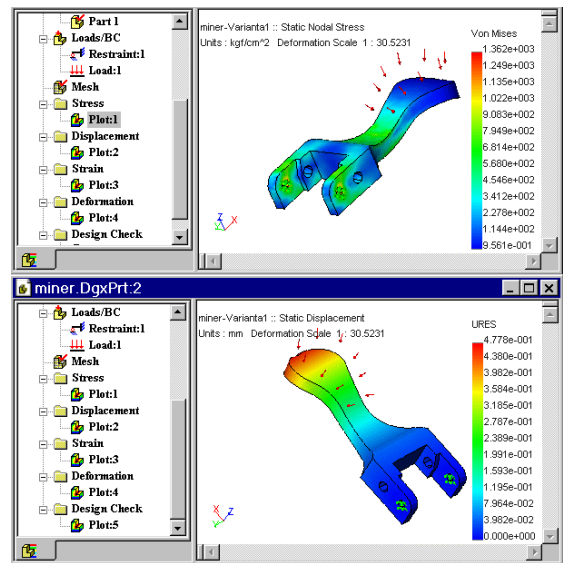

Figure 13.

REFERENCES

[1] Mănescu, T.Ş, Câmpian, V., Nedelcu, D., Pinca B.,C., A Vane Analysis by Finite Element Model, The VIIth International Symposium Interdisciplinary Regional Research ISIRR 2003, section V, pp. 500-505, Hunedoara, România

[2] Structural Research and Analisys Corp - COSMOS Design Star Product – Nonlinear Stress Analisys User's Guide and Tutorial, Los Angeles, 2001.