5. POST-PROCESSING PHASE

Post-processing is the part of the analysis process that involves reviewing and interpreting the results from the solver. Back in the 60's the specialists at NASA would have just got reams of paper from their dot matrix printers, full of numbers that they'd have to manually review to figure understand the analysis results.

But again, thanks to modern computers and software, the post processing software now offers the engineer nice colouredpictures on the screen to show the deformed shape of the part and where any stress 'hot spots' may be [10].

Whilst this may seem a little gimmicky, these coloured pictures, technically known as contours, are a very intuitive way of interpreting the results and quickly getting a practical picture of the overall state of the part, regardless of your technical knowledge… anyone can see if it's red, that's generally bad! The post processor will also show you the deformed shape which helps the analyst understand how the stresses are developing and what changes can be made to improve the design.

Although modern post processors makes viewing results quite straight forward, accurate interpretation of the analysis still requires a thorough knowledge of engineering principles, stresses and material properties. A good analyst will know what changes need to be made to the part to reduce areas of high stress and even determine how much material can be removed from areas of low stress, resulting in a stronger, lighter part [10].

5.1. About pre- and post-processor

The greatest challenge for engineering managers is mitigating the risks inherent in any new product design. FEA technologies help enable significant reductions in risk, which is why they are so widely used. Industry leading pre- and postprocessing can provide an additional order-of-magnitude gain in analyses. The gains are in accuracy and control – simplifying and cleaning up the geometry and the discretized data that goes into creating the finite element model and ensuring that the calculated results are both understandable and relevant. Fig. 137 illustrates the role of the pre- and postprocessor and solver in the product design process [50].

The main problem of post-processing phase is to understanding the results. Given the potentially huge amount of data that can be created by a finite element solution, it's important to be able to quickly interpret results and gain an understanding of the model's behavior for a quick analysis turnaround [50].

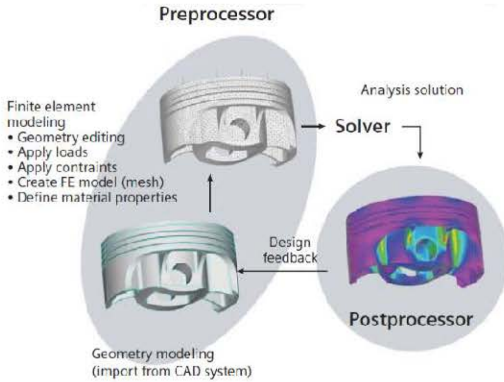


Fig. 137 The finite element analysis process [50]

The role of the preprocessor, as seen in Fig. 138, is to import the geometry data, correct the geometry and discretize or mesh it in order to idealize a physical design and create an FE model for analysis. Automation and customization capabilities provided by the preprocessor can help to speed up this process. Following analysis by a solver, the postprocessor imports and displays the results in a graphical format and helps the understanding of model behavior. With a good understanding of the performance of the design from the analysis, the analyst may return to the preprocessor to further refine the model if necessary, and re-run the analysis [50].

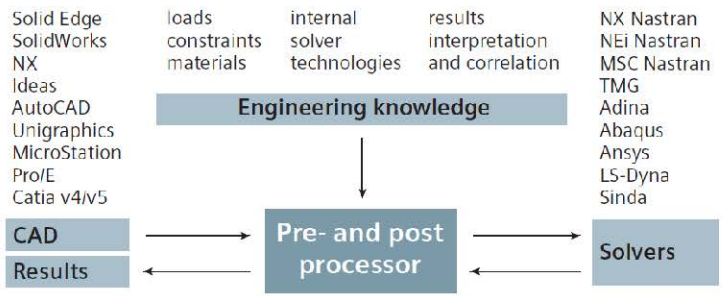


Fig. 138 The role of the pre- and the postprocessor [50]

Each time an FE model is solved, it can help create a vast amount of results data. The ability to process the data and quickly gain an understanding of the model behavior is important for a fast analysis turnaround. The postprocessor should therefore allow full control of results selection and include a robust and varied set of tools to manage and display results, while at the same time facilitate easy comprehension of the data. Results viewing becomes more complex with highly idealized models, so the postprocessing tools should provide the ability to easily view appropriate results quantities on shell and beam elements [50].

5.2. General considerations about the post-processing phase

After a finite element model has been prepared and checked, boundary conditions have been applied, and the model has been solved, it is time to investigate the results of the analysis. This activity is known as the post-processing phase of the finite element method. Post-processing begins with a thorough check for problems that may have occurred during solution. Most solvers provide a log file, which should be searched for warnings or errors, and which will also provide a quantitative measure of how well-behaved the numerical procedures were during solution.

Next, reaction loads at restrained nodes should be summed and examined as a "sanity check". Reaction loads that do not closely balance the applied load resultant for a linear static analysis should cast doubt on the validity of other results. Error norms such as strain energy density and stress deviation among adjacent elements might be looked at next, but for h-code analyses these quantities are best used to target subsequent adaptive remeshing [47].

Once the solution is verified to be free of numerical problems, the quantities of interest may be examined. Many display options are available, the choice of which depends on the mathematical form of the quantity as well as its physical meaning. For example, the displacement of a solid linear brick element's node is a 3-component spatial vector, and the model's overall displacement is often displayed by superposing the deformed shape over the undeformed shape.

Dynamic viewing and animation capabilities aid greatly in obtaining an understanding of the deformation pattern. Stresses, being tensor quantities, currently lack a good single visualization technique, and thus derived stress quantities are extracted and displayed. Principal stress vectors may be displayed as color-coded arrows, indicating both direction and magnitude. The magnitude of principal stresses or of a scalar failure stress such as the Von Mises stress may be displayed on the model as colored bands. When this type of display is treated as a 3D object subjected to light sources, the resulting image is known as a shaded image stress plot. Displacement magnitude may also be displayed by colored bands, but this can lead to misinterpretation as a stress plot [47].

An area of post-processing that is rapidly gaining popularity is that of adaptive remeshing. Error norms such as strain energy density are used to remesh the model, placing a denser mesh in regions needing improvement and a coarser mesh in areas of overkill. Adaptivity requires an associative link between the model and the underlying CAD geometry, and works best if boundary conditions may be applied directly to the geometry, as well. Adaptive remeshing is a recent demonstration of the iterative nature of h-code analysis [47].

Optimization is another area enjoying recent advancement. Based on the values of various results, the model is modified automatically in an attempt to satisfy certain performance criteria and is solved again. The process iterates until some convergence criterion is met. In its scalar form, optimization modifies beam cross-sectional properties, thin shell thicknesses and/or material properties in an attempt to meet maximum stress constraints, maximum deflection constraints, and/or vibrational frequency constraints.

Shape optimization is more complex, with the actual 3D model boundaries being modified. This is best accomplished by using the driving dimensions as optimization parameters, but mesh quality at each iteration can be a concern [47].

Another direction clearly visible in the finite element field is the integration of FEA packages with so-called "mechanism" packages, which analyze motion and forces of large-displacement multi-body systems.

A long-term goal would be real-time computation and display of displacements and stresses in a multi-body system undergoing large displacement motion, with frictional effects and fluid flow taken into account when necessary. It is difficult to estimate the increase in computing power necessary to accomplish this feat, but 2 or 3 orders of magnitude is probably close. Algorithms to integrate these fields of analysis may be expected to follow the computing power increases [47].

In summary, the finite element method is a relatively recent discipline that has quickly become a mature method, especially for structural and thermal analysis. The costs of applying this technology to everyday design tasks have been dropping, while the capabilities delivered by the method expand constantly. With education in the technique and in the commercial software packages becoming more and more available, the question has moved from "Why apply FEA?" to "Why not?". The method is fully capable of delivering higher quality products in a shorter design cycle with a reduced chance of field failure, provided it is applied by a capable analyst. It is also a valid indication of thorough design practices, should an unexpected litigation crop up. The time is now for industry to make greater use of this and other analysis techniques [47].

5.3. Results

The primary results in a finite element analysis are grid point displacements and rotations. Element results such as stresses, strains, and strain energy density are derived from those results. Other results include element forces, MPC forces, SPC forces, and grid point forces. Results of a finite element analysis are post-processed using a graphical tool [4].

5.3.1. Displacements [4]

Displacements and rotations are computed in linear static, and frequency response analyses. In addition, in frequency response velocities and acceleration are computed.

Eigenvectors are the primary result in a normal modes and buckling analyses. In a normal modes analysis, they are normalized with respect to the mass matrix or with respect to the maximum vector component. In a buckling analysis, the latter always applies.

Displacements, velocities, accelerations, and eigenvectors are grid point results. They are plotted as a deformed structure, or as a contour on the undeformed structure. Some post-processors, such as Ansys or Catia, also allow the animation of the displacements [4].

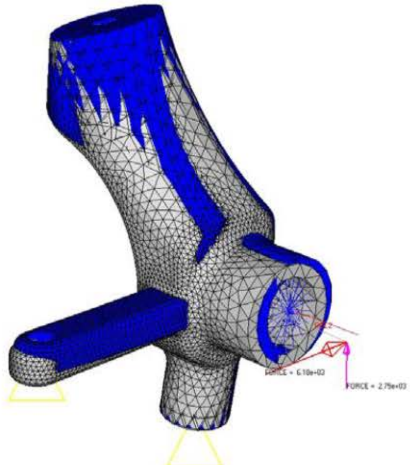


Fig. 139 Original and deformed displacement contour plot [10]

5.3.2. Stresses [4]

The stresses are secondary results in a static analysis.

Stresses near notches and other sharp corners, point loads and boundary conditions, and rigid elements are often unreliable due to the singularities in these points. A mesh refinement in such places can improve the stress prediction. A theoretically infinite stress cannot be predicted by finite elements.

Stresses are primarily calculated at the Gauss integration points. These give the most accurate prediction. However, element stresses, corner stresses, and grid point stresses are provided.

Element stresses are calculated at the centroid of the element. They should only be post-processed using an assign plot. Contouring of element stresses vastly underestimates the extreme values due to the smearing across element boundaries.

The stresses of interest are usually found on the surface of a structure. Mesh refinement will actually not just improve the stress prediction but also change the location of the point of stress evaluation. Therefore, it is common practice to use a skin of thin membrane elements in 3D modeling, or rod elements in 2D modeling, to evaluate the stresses on element surfaces or edges, respectively. This method is accurate since it considers the correct condition of a stress-free boundary if no load is applied to the boundary. The method of skinning a model also has the advantage of much faster post-processing of solid models because only the membrane skin needs to be displayed.

Besides assign plots, elements stresses can be viewed in tensor plots that can help in the evaluation of the load path in a structure by evaluating the principal stress directions. Corner stresses are computed by extrapolating the stresses from the Gauss points to the element grid points. Corner stresses are plotted in a contour plot. Corner stresses for solid elements are not available for normal modes analysis.

Grid point stresses are computed by averaging the corner stresses contributions of the elements meeting in a grid point. The averaging does not consider the condition of a stress-free boundary. Further, interfaces between different materials, where a stress jump normally can be observed, are not considered correctly because of the smearing of the stress. Grid point stresses are plotted in a contour plot. For first order elements, grid point stresses do not provide higher accuracy over element stresses. For second order elements, the stress prediction might improve by using grid point stress over element stresses, considering the weaknesses mentioned above [4].

The next step is to determine whether a part will break by comparing the stress values from the analysis results to the strength of the material. Every metal and most plastics have what's called a yield strength and an ultimate strength. If the stress within the part exceeds the material yield strength, then the part will not return to it's original shape when the load is removed. Although the part is still in one piece, it's going to remain bent, which generally isn't good. If the stress exceeds the ultimate strength, then the part will fracture and break. Ideally, the whole aim of the analysis is to make sure the stresses within the part remain below the yield strength of the material [10].

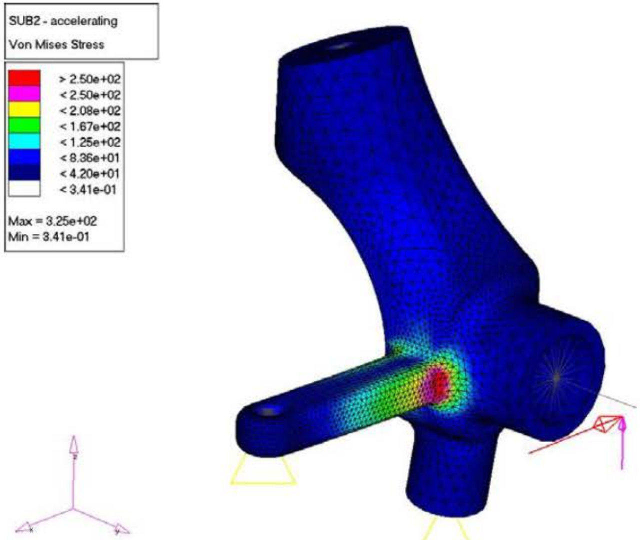


Fig. 140 Stress Contour (units are MPa) [10]

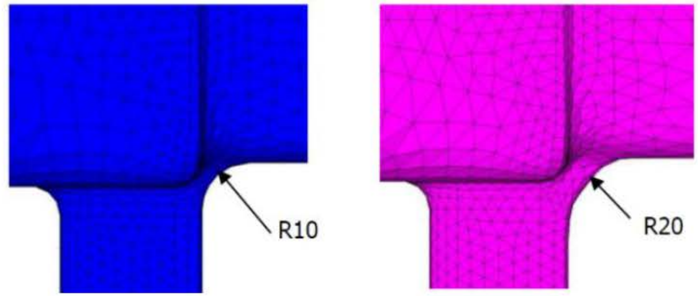


Fig. 141 Increased Fillet Radius [10]

In our suspension upright example, the stress contour shows a maximum stress of 325 MPa, which is above the material's yield strength of 250 MPa, but below the tensile strength of 345 MPa. Sothis means the part will bend under these loads, but won't actually break in two. Still, a bent upright is of no use to anyone so one possible change to make to the part would be to increase the radius of the fillet in the high stress area to add some extra material.

And after rerunning the analysis, it's clear that the maximum stress in the part has dropped to a much more acceptable value of 120 MPa. Much better to have figured this out now, rather than after having parts made and tested (not to mention cheaper and safer!).

It’s clear that there are many benefits to using this type of simulation tool in engineering: reduced costs; reduced design time; being able to assess a wide variety of designs; and ending up with a stronger, lighter part.

5.3.3. Strains [4]

Strains are secondary results. They are calculated as elements strains. Remarks made above on element stresses apply here too.

5.3.3.1. Strain Energy Densities

Strain energy densities are secondary results in static and normal modes analysis. They are calculated as element strain energy densities. Remarks made above on element stresses apply here too.

5.3.3.2. Forces

Element forces, MPC forces, SPC forces, and grid point forces are printed as tabulated output.

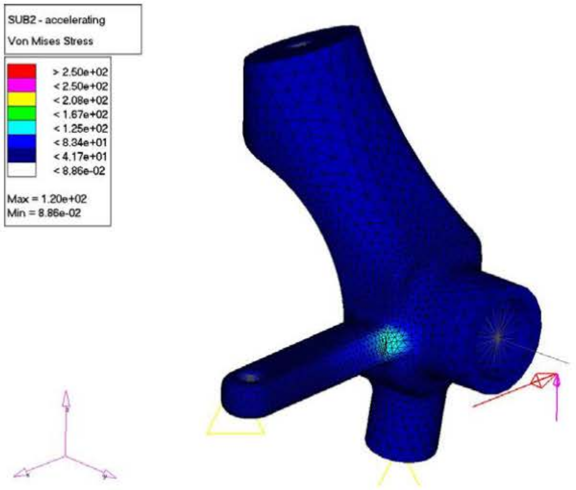


Fig. 142 Results for Revised Design [10]

5.4. Samples of post-processing

5.4.1. Post-processing in CATIA [14]

The **case study** for this lesson is the Drill Press Table FE model post-processing (Fig. 143). The focus of this case study is to post-process the computed data as per requirement.
You must decide which results are required to be viewed. Based on this decision you will create the visualization images and an analysis report which will clearly summarize the results of the computation.

- You will create Translational Displacement Magnitude Plot, also find the value of maximum displacement and its location (Fig. 144).
- You will create plots for Maximum and Minimum Principle stress distribution with discontinuous iso. Find out the location of the maximum values.
- You will create Von Mises stress distribution plot and find the location of maximum value in FE model.
- You will create Local error distribution and find the location of maximum value.
- You will create output sensors for Error in Energy, Global Error Rate percentage.
- You will create Resultant sensor to know reactions at restraints.
- You will create Analysis Report with Images.

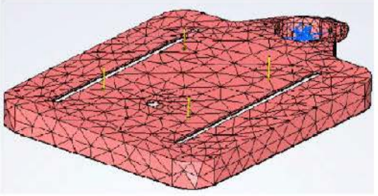


Fig. 143 Drill press Table FE Model

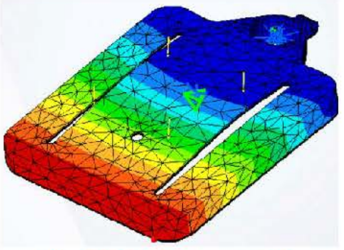


Fig. 144 Drill press Table Translational Displacement Image

Stages in the Process

This will involve the following steps to perform the case study. First, you will learn what is meant by Post-processing. Later, you will see how to use the various result visualization functionalities in the GPS workbench.

1. Understand Post-processing.

2. Create Visualization Images.
3. Create Extrema on created images.
4. Use sensors.
5. Generate analysis report.

What is Post-processing

Post-processing involves all those steps which are carried out after computation of results:

- Create different images like Deformation, Displacement magnitude, Stress, Reaction
- force, and other available images from computed solution data.
- Find location of result values in FE model.
- Validate the results using different images and study these images to understand and interpret the solution.
- Make decisions for further improving the solution with mesh refinement iterations or other solution types.
- Validate the current design or provide the changes based on the results.

Visualization Images

CATIA provides the different visualization images to study the results of the analysis as shown in the table below. When you edit an image, you can choose the image type. Location of the result values, provided by the solver, changes according to the selected type. You can choose the location for text and symbol types only.

Image	Purpose of the image
Deformation	To visualize finite element mesh in the deformed configuration
Displacement	To visualize displacement field patterns which represent variation of position vectors of material particles
Principal Stresses	To visualize principal stress field patterns which represent a tensor field quantity used to measure the state of stress
Von Mises Stresses	To visualize the Von Mises Stress field patterns, which represent a scalar field quantity obtained from the volume distortion energy density
Precision	To visualize computation error maps

Sensors

Sensor is a physical output of a computation. Sensors allow you to produce specialized output provided as a single value rather than a range of values displayed on the model.

You can use sensor output to:

- Validate the Analysis Results
- Synthesize the analysis results and use it as a parameter to improve and optimize design specifications.

Global sensor (Fig. 145) provides an output value for the entire FE model, while local sensor provides an output value for a local region in the FE model.

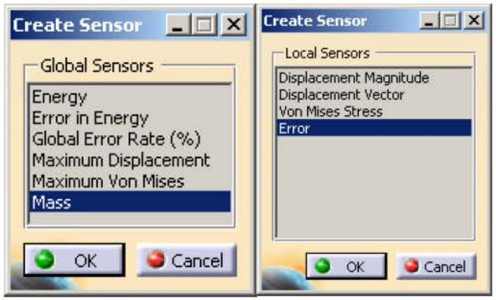


Fig. 145 Sensors

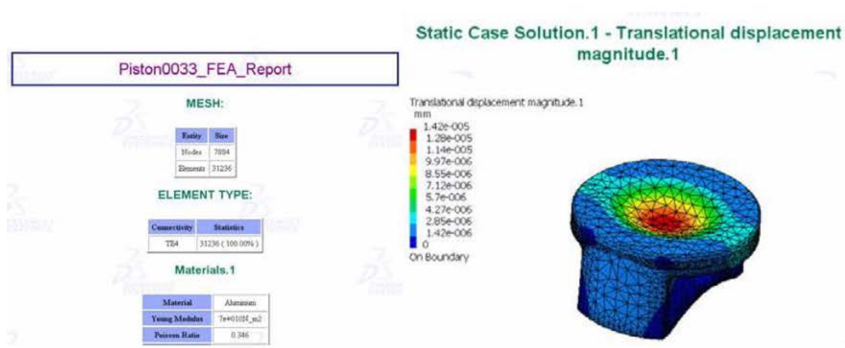
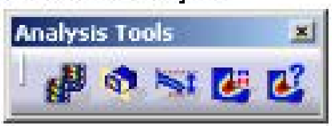


Fig. 146 Report

Report Generation

After the required level of solution accuracy is reached, after and several mesh refinement iterations, you can generate reports. In this step of report generation, you will present the information generated during all stages of the FE Analysis process: Pre-processing, Computation, and Post-processing (Fig. 146).

Main Tools

Customize View Parameters 	Lets you customize the view mode.
Deformation	Lets you specify the finite element mesh in the deformed configuration under applied loading condition.
Displacement 	Lets you visualize the displacement field patterns, which represent variation of position vectors of material particles as a result of applied load
Von Mises stress	Lets you visualize the Von Mises Stress field pattern.
Principle stress 	Lets you visualize the principle stress field patterns which represent a tensor field quantity used to measure the state of stress and to determine the load path on loaded part
Precision	Lets you visualize the computation error maps.
Cut Plane Analysis 	Lets you to visualize sections of the structure to allow you to visualize the results within the material
Generate Reports 	Lets you to generates report for computed solutions.

5.4.2. Post-processing in ANSYS [6]

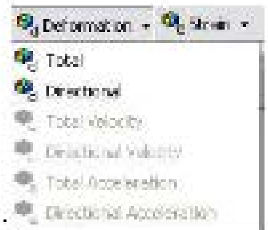
Numerous structural results are available:

- Directional and total deformation.
- Components, principal, or invariants of stresses and strains.
- Contact output.
- Reaction forces.
- More ....

In Mechanical, results may be requested before or after solving.

- If you solve a model then request results afterwards, click on the "Solve" button, and the results will be retrieved (the results file is re - read).
- You can also right click the Solution branch or a new result item and "Evaluate All Results". 
- A new solution is not required.

Contour and vector plots are usually shown on the deformed geometry. Use the Context Toolbar to change settings.



Results can be scoped to nodes using named selections. The deformation of the model can be plotted:

- Total deformation is a scalar quantity:

$$U_{total} = \sqrt{U_x^2 + U_y^2 + U_z^2}$$

- The x, y, and z components of deformation can be requested under “Directional”, in global or local coordinates.
- Vector plots of deformation are available (see Fig. 148).

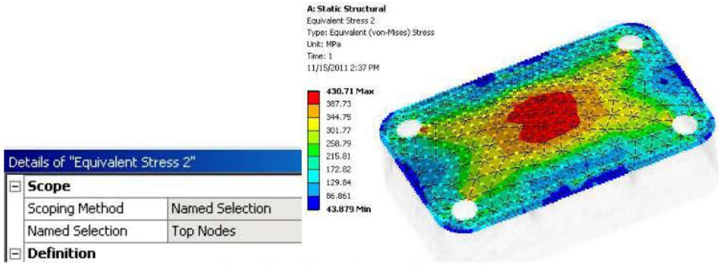


Fig. 147 Details of Equivalent stress

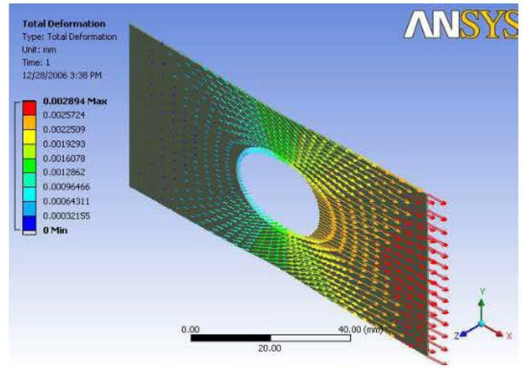
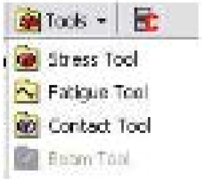


Fig. 148 Vector plot

Stresses and strains:

- Stresses and (elastic) strains have six components (x, y, z, xy, yz, xz) while thermal strains have three components (x, y, z)
- For stresses and strains (Fig. 149), components can be requested under “Normal” (x, y, z) and “Shear” (xy, yz, xz). For thermal strains, (x, y, z) components are under “Thermal.”
- Principal stresses are always arranged such that s1 > s2 > s3
- Intensity is defined as the largest of the absolute values

  – s1 – s2, s2 – s3 or s3 – s1



Stress Tool:

Calculates safety factors based on several material failure theories:

- Ductile Theories:
  - Maximum Equivalent Stress
  - Maximum Shear Stress
- Brittle Theories:
  - Mohr - Coulomb Stress
  - Maximum Tensile Stress
- Safety factor, safety margin and stress ratio can be plotted.

- User specified failure criteria can be entered.

Contact results are requested via a "Contact Tool" under the Solution branch. Contact regions can be selected in the graphics window or using a Worksheet (Fig. 150).

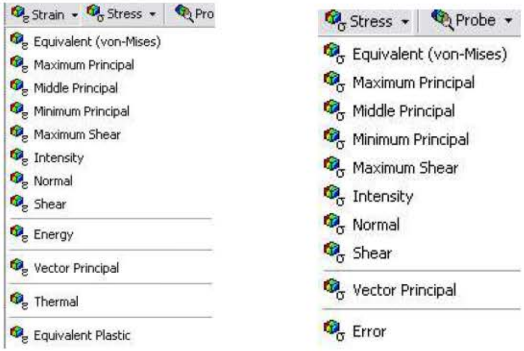


Fig. 149 Menu for stresses and strains

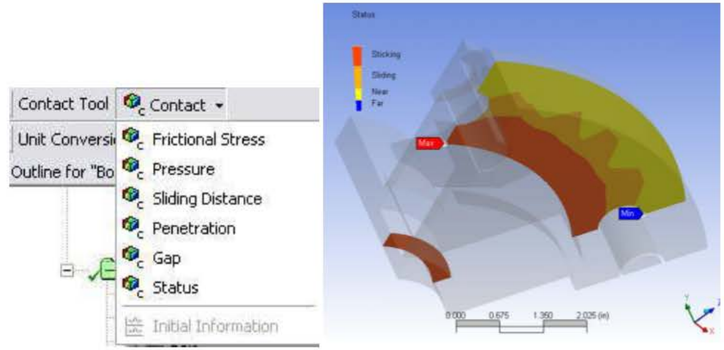
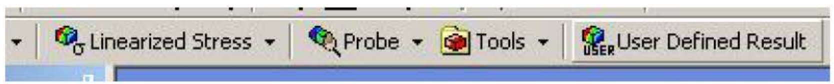


Fig. 150 Contact tool and contact results

In addition to the standard result items one can insert "**user defined**" results. These results can include mathematical expressions and can be combinations of multiple result items. Define in 2 ways:

- Select "User Defined Result" from the solution context menu



- OR – From the Solution Worksheet highlight result > RMB > Create User Defined Result.

User Defined Result Expressions

Type	Data Type	Data Style	Component	Expression	Output Unit
U	Nodal	Scalar	X	UX	Displacement
U	Nodal	Scalar	Y	UY	Displacement
U	Nodal	Scalar	Z	UZ	Displacement
U	Nodal	Scalar	SUM	USUM	Displacement
U	Nodal	Vector	VECTORS	UVECTORS	Displacement
S	Element Nodal	Scalar	X	SX	Stress
S	Element Nodal	Scalar	Y	SY	Stress
S	Element Nodal	Scalar	Z	SZ	Stress