

TRIDENT FEA 2004

Release Notes

Model Generation/Modification

The modeling feature in Trident provides the user with the capability to generate entire finite element models, including elements and properties, boundary conditions, lumped masses and loads. In the 2004 release, several new capabilities have been provided.






- **Mapping Nodes to a Sphere:** Active nodes can be automatically projected onto a spherical surface, by defining the radius and centre of the sphere.
- **Corrosion Allowance:** The thickness of plate, web or flange elements can be reduced by means of an element corrosion allowance, which is based on a thickness or a percentage specified by the user.
- **Meshing of “End Caps”:** A mesh for a flat or domed end cap can be created by specifying the number of elements in the radial and circumferential directions, the outer radius and the dome radius.
- **Nonlinear Material Property:** A new capability is provided to allow the user to create a nonlinear material property or to define a formulation property for a nonlinear material, by specifying isotropic hardening, kinematic hardening, or a nonlinear stress-strain curve (i.e., a multi-linear stress-strain curve defined by up to 20 pairs of stress-strain values). The user can verify nonlinear material properties by means of a summary table that displays the number of elements assigned a nonlinear material, or a nonlinear geometry and material. The user can also generate X-Y plots of the nonlinear material stress-strain curve that has been created (or one that is contained in the nonlinear material property database or assigned to an individual element via the formulation operation).
- **New Methods for Lumped Masses:** In addition to the manual method and the Weight Curve method of creating lumped masses, two new methods are now provided: reading a “mass coordinate table” or a “mass volume table” from a file. The user can shift the coordinates or apply a conversion factor to the coordinate or mass data.
- **Global Model:** The user can now “unassign” a global model, in situations where a finite element model was previously assigned to a global model status (for global/local analysis), but a decision is subsequently made not to use the finite element model as a global model.

Load Generation

- **Global Summations:** A capability has been provided for displaying a table showing the summation of forces and moments in each direction for pressure loads, concentrated loads, inertia loads, and all loads combined. This capability allows the user to verify the applied load case.

- **Inertia Loads:** An enhanced capability is provided for handling inertia loads. Previously, inertial loads were computed using material densities by default. The user now has the option to choose whether element and/or lumped masses are to be used in generating inertia loads, and whether element densities are to be based on material densities or element mass load densities. The user can also choose to generate inertia loads only, or to generate no inertia loads.
- **Static Balance of Loads:** In the Standard Method for static balance of loads, the computation of artificial acceleration has been improved by including forces due to motions. The bending moment and shear force table generated by the static balance of loads can be used to generate an X-Y plot of bending moment and shear force diagrams.

Model Display

- **Full Screen/ Expanded Mode:** This capability is activated through the  button on the Autoview menu located on the far left hand side of the Viewport. The button toggles between Full Screen Expanded mode  and the normal mode .
- **Viewport Capabilities:** Various capabilities related to the Viewport have been enhanced:
The  (negative perspective) button has been modified to toggle between positive and negative, to indicate whether the positive or negative perspective will be displayed when the user presses the button.
The  (maximize) button displays the model in the Viewport and the space normally reserved for plot legends. However, in this Full-Screen Expanded mode, the submenus and plot legends (such as the Verify colour bar) often overlap the displayed model. These problems have been resolved by reducing the space taken up by the plot legends and providing the capability to drag the submenus across the screen.
- **Multiple Viewports:** Previously, the user defined the locations of the multiple viewports. This capability has been enhanced to make it more robust, by providing several predefined locations in the upper left, upper right, lower left or lower right quarters of the Viewport. If more space is required, larger viewports can be defined using the upper, lower, left or right halves of the Viewport.
- **Animation Pause:** When displaying animation files, the pause capability now includes “manual” and “auto” settings that have the program pause manually or automatically (at the end of each sequence). In pause mode, sequencing is continued by clicking the left mouse button, or terminated by clicking the right mouse button.

Program Options and Files

- **User-Specified Unit Conversions:** Previously, the user could convert data values to a new system of units by specifying the old and new units for length and force. This capability has now been enhanced to allow the user to define conversion factors for the length and force.
- **Enforced Displacements:** A new capability is provided for a top down analysis of a local model. The prescribed displacements from a global analysis have been

previously stored in a “*prefix.dis*” file using node numbers. Trident FEA has been enhanced so the user can now save the data as enforced displacements in a “*prefix.edf*” file using coordinates. This permits them to be reused even if the node numbering in the model has changed. When the prescribed displacements from the “*prefix.edf*” file are read into the database, a “*prefix.dis*” file is created for the analysis. The capability has been provided so the user can verify the data before running the solver on the local model by plotting the prescribed displacements.

- **New Auxiliary File for Active Elements:** A new auxiliary file that stores active elements can be used for display. A desired view can be stored to eliminate the repeated use of the Erase/Restore feature to obtain the same view.
- **Option to Delete Temporary Files created by Solver:** Filename extensions have been redefined, so that temporary files created by the solver can be automatically deleted. The user specifies if a non-linear restart, full restart or no restart is required.

Results

- **Deformations:** An enhanced capability is provided for applying displacements or mode shapes from a file to create deformations of a model. Previously, the node coordinates were modified by applying the translational components of the deformations to the coordinates of the currently active nodes. In the past, the user had to simply accept this load case. The user can now apply independent scaling factors to the X, Y and Z directions, to generate many variations of the load case