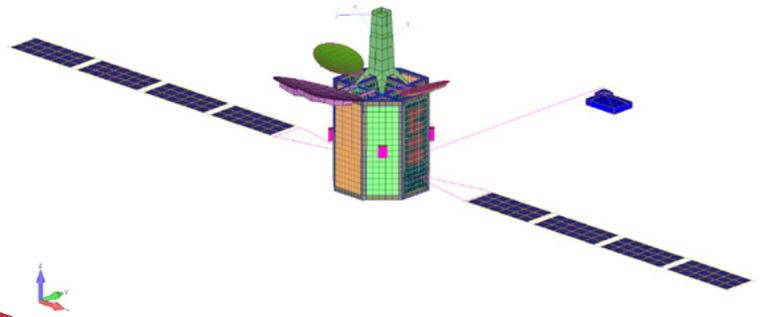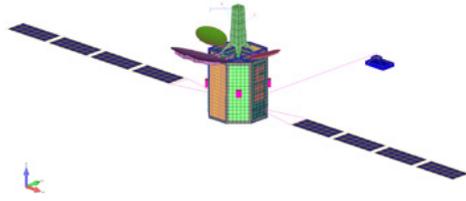Femap 11.1

Group Functionality in Femap

Group Functionality in Femap

Software:
Femap 11.1

Overview

The Groups ability in Femap can be used for organizing complex models. Groups, which can be used while building the model and while viewing results, can be used to view models with nested parts, with several mesh layers, and by separate parts, and they can store geometry, mesh, loads, and constraint items. This document explains how to create groups and gives helpful tips for using groups.

This whitepaper is part of a series of free Siemens PLM Software training resources provided by ATA. For more whitepapers, tutorials, videos, and macros, visit ATA's PLM Software website: http://www.ata-plmsoftware.com/resources.

Group Functionality in Femap

Group Overview

Geometry and finite element models are often very complex, making it impractical to view the entire model at once. Organizing the model into groups by geometry or FEM entities allows the user to view each section of the model separately.

Groups are listed in the Groups dropdown list in the Model Info tree. There are three viewing options that can be accessed by right-clicking anywhere in the Groups section of Model Info: Full Model, Active Group, and Multiple Groups.

The Model Info tree is shown in Figure 1, with three groups in the model expanded under the Groups heading. The active group is indicated by blue shading of the group title—in this case, "Group 1."

Figure 1: ▶
Model Info pane.
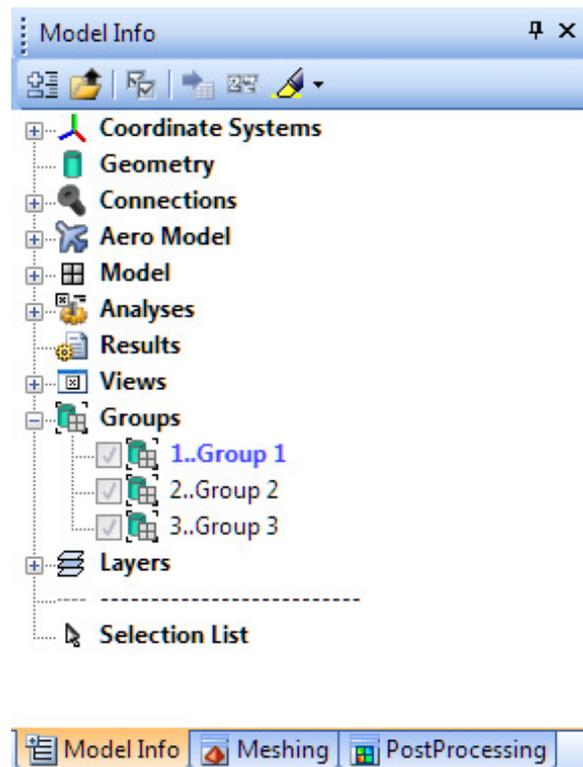
Once groups are created, a group can be made active by either double-clicking the group name or right-clicking the group and selecting Activate. When the Show Active Group option is selected, this group will be shown in the graphics window. The ID of the active group can be seen in the view legend, if visible, or in the active item list in the Status bar (at the lower right-hand corner of the screen).

Group Functionality in Femap

Figure 2: ▶
Group viewing options.
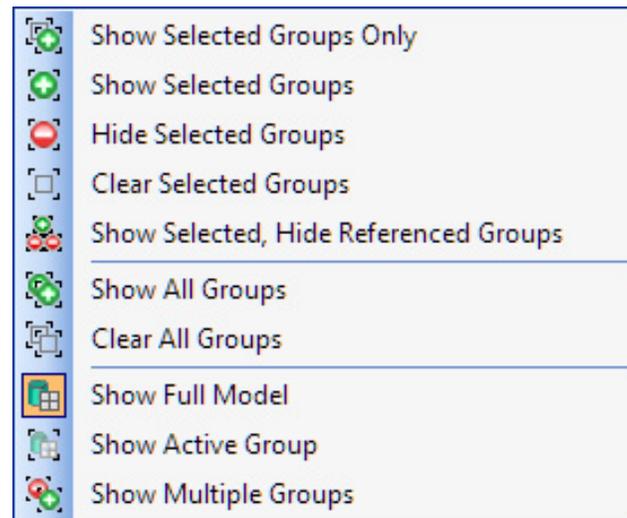
There are also several options for viewing multiple groups simultaneously. If multiple individual (Ctrl) or a range (Shift) of group titles are selected, the user can right-click and select Show Multiple Groups. If Show Multiple Groups is already selected, the checkboxes next to each group title become selectable. Groups can be shown or hidden by left-clicking in those checkboxes. In addition, action can be taken across multiple groups by right-clicking on the checkboxes and selecting one of the options shown in Figure 2.

Group Creation

Groups can be generated automatically by solid, property, material, or element type. This is useful for quickly creating groups when the model is already well organized by one of these categories. This option can be accessed under Group → Operations → Generate*.

A new group can also be created manually, by right-clicking in the Group section of the Model Info pane and selecting New, or by selecting Create/Manage and New Group under the Group drop-down menu in the Menu toolbar. A newly created group will automatically become the active group.

There are dozens of methods for adding entities to a group. Entities can be added to groups by ID by entering specific element numbers, or by pasting in a list (using the Pick^ button in the Entity Selection dialog box). Entities can also be added by selecting related entities (e.g., all elements that are defined by a specific property can be added to a group by selecting that property). In Figure 3, using Group → Element → Property, the brown-colored property is selected, and all elements defined by that property are added to the active group.

     Femap is a trademark of Siemens PLM Software, Inc.
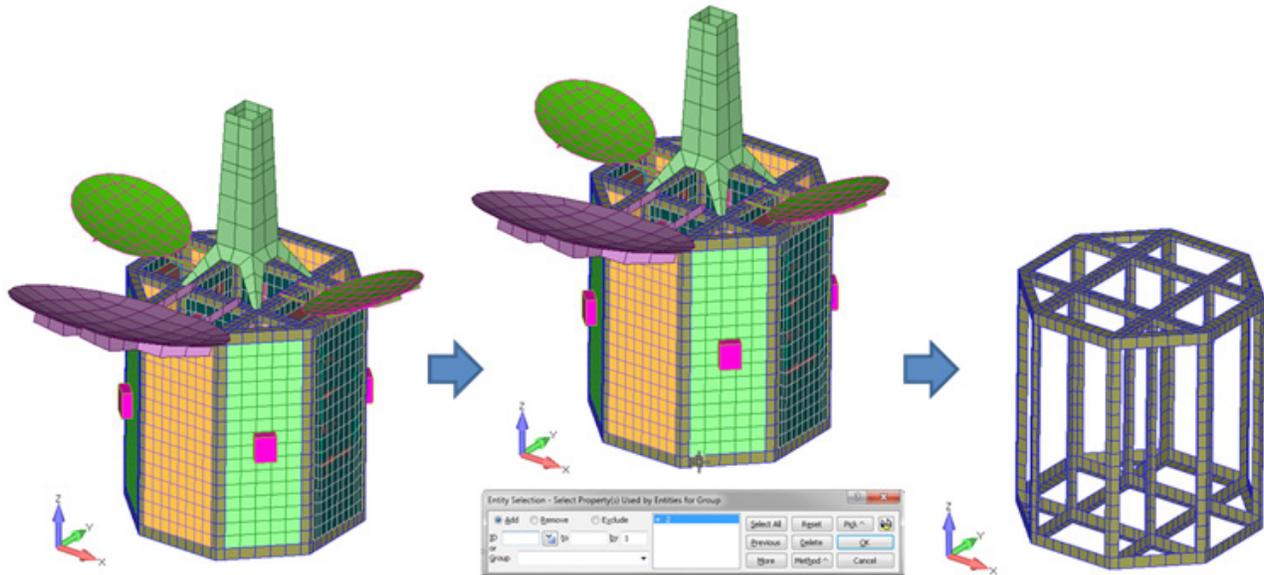
Group Functionality in Femap

Figure 3: ▲
Creating an element group by property.

Manual grouping is also a useful way to view entities related to an area of interest. For example, your model may be failing quality checks on a specific element. That element can be viewed alone by creating an empty group and adding that element to the group by ID. Nodes associated with that element can then be added to the group (Group → Node → On Element), and then elements associated with those nodes can also be added (Group → Element → Using Node).

Group Boolean Operations

Boolean operations are available with Femap groups. This can be used to combine groups, subtract one group from another, or determine what groups have or do not have in common. These operations are available under Group → Operations → Booleans. A new group is created any time a Boolean operation is performed, identified by ID and title at the top of the Group Booleans dialog box. Figure 4 uses this functionality to find the elements in common between groups 1 and 2. The "In All" option is selected, and the group is created as ID 4. The image of the newly created group 4 identifies the elements in common between groups 1 and 2.
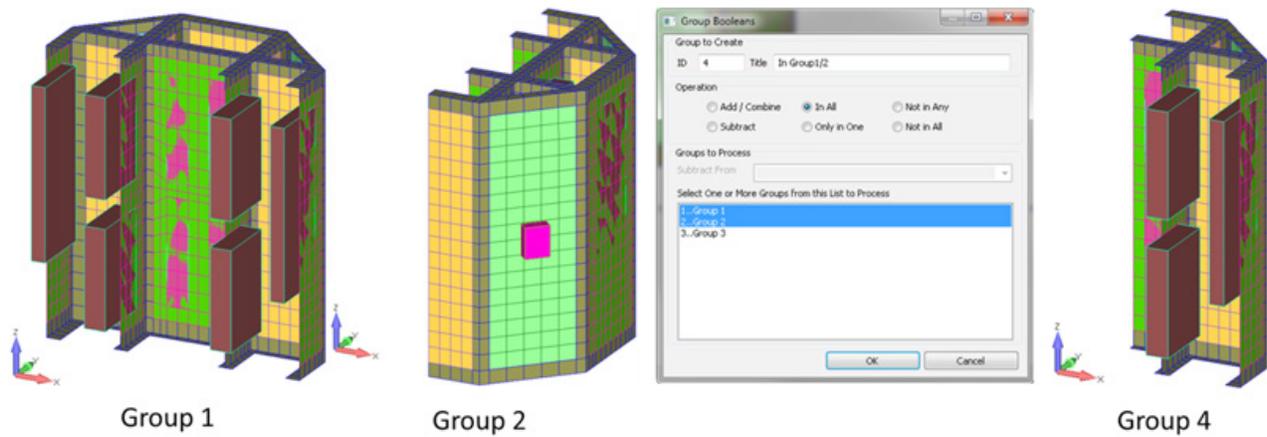
Group Functionality in Femap

Figure 4: ▲
Using Group Boolean operations.

General Notes for Using Groups

Groups can easily be renumbered or rearranged either individually or several at a time. A single group can be renumbered by right-clicking that group and choosing Renumber. Several groups can be renumbered under Modify → Renumber → Group and entering their IDs.

Groups can be created automatically when importing a FEM from a Nastran deck that uses "include" statements to combine multiple files. Go to File → Preferences → Interfaces tab and check "Create Groups from Include Files" under the Nastran Solver Write Options section. Then when the FEM is imported into Femap, all the entities from a given file (coordinate systems, nodes, elements, materials, and properties) included in the deck will automatically be assigned to a new group. The name of the file will be assigned as the group title.

www.ata-plmsoftware.com

ATA Engineering

Solution Partner

PLM

SIEMENS

🏠 www.ata-e.com

in ata-engineering

🐦 @ataengineering

✉ sales@ata-e.com

📞 858.480.2000

www.ata-plmsoftware.com

San Diego
Corporate Headquarters   Albuquerque   Denver   Huntsville   Los Angeles   San Francisco   Washington, D.C.