Altair HyperView™

Free Body Diagrams (FBD)

The Free Body Diagram (FBD) tool facilitates the extraction and post-processing of Grid Point Force (GPFORCE) results and can be used create and edit Free Body Diagrams (FBD).

FBD extractions are typically utilized for breakout and/or sub-modeling analysis schemes, where balanced "free body" sub-cases are extracted from a coarse grid model and applied to a fine grid sub-model for eventual optimization and/or analysis. FBD is also used to extract section resultant forces and moments (typically at the centroid of a section) for use in traditional strength calculations.
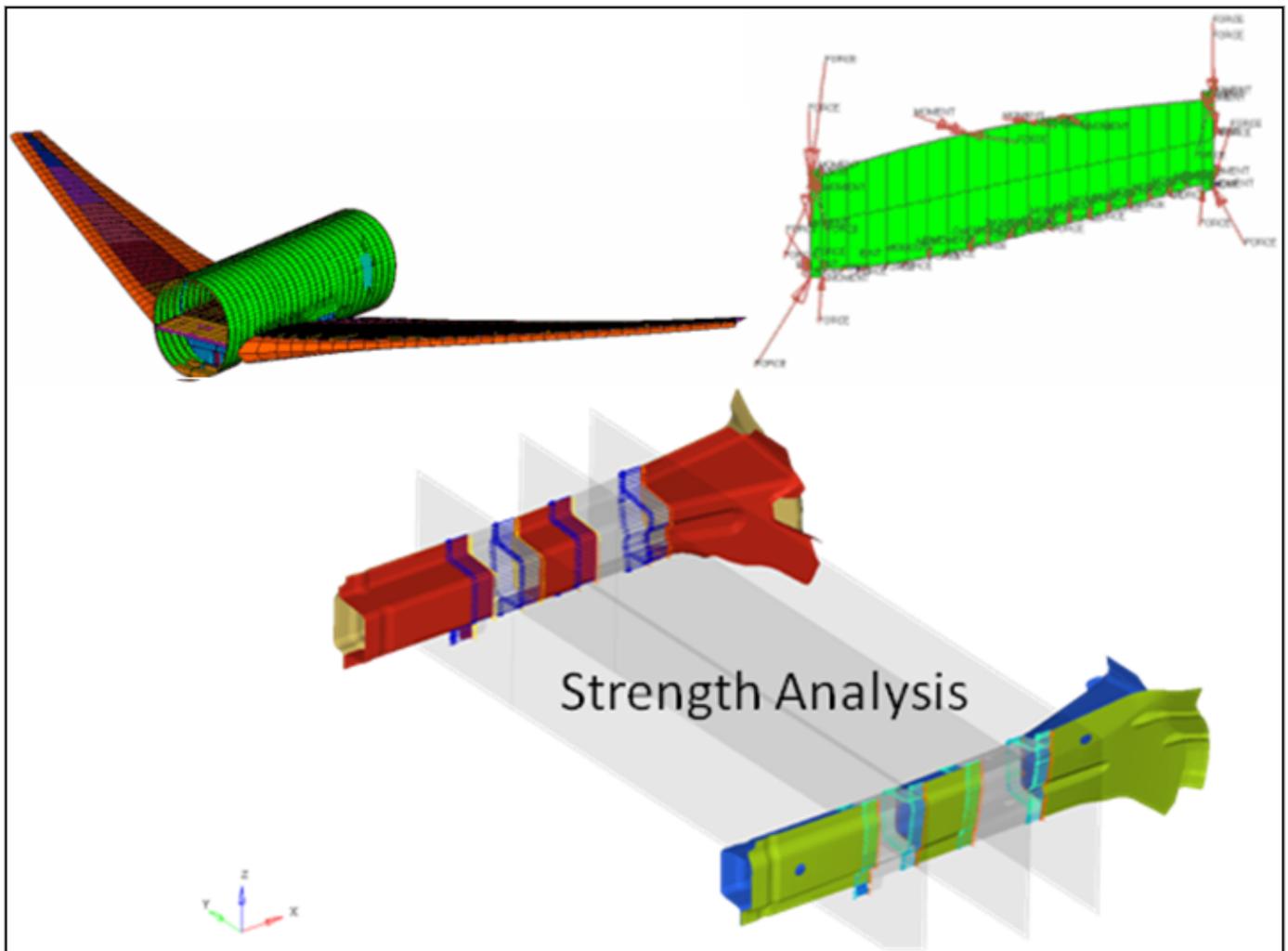

Figure 1.

FBD Utility

The FBD utility extracts grid point force data (including forces and moments) for a user-defined element set and is useful for doing breakout modeling within a sub-modeling scheme. Results can be output for graphical review, a text summary table, and/or a formatted Comma-Separated Values (`.csv`) file which can be loaded into traditional spreadsheet software packages.

The data from the currently loaded model/results file and the selected load case/simulation in the Results browser will be used for all grid point force based calculations. The FBD utility currently supports ANSYS (Rst), Nastran (Op2/XDB), OptiStruct (Op2), Abaqus (ODB), and H3D result files.

Note: The following data types must be present in the result file to launch and use the FBD tool:

- ANSYS: "Element Nodal Force as Vector" and "Element Nodal Moment as Vector"
- Nastran/OptiStruct: "Grid Point Element Forces" and "Grid Point Element Moments"
- In the case of OptiStruct H3D, make sure that GPFORCE(FBD) is requested for output when setting up the analysis.
- Abaqus: NFORC Forces/Moments

Important: The Advanced Result Math template must be selected when loading the model and result files in the Load Model panel.
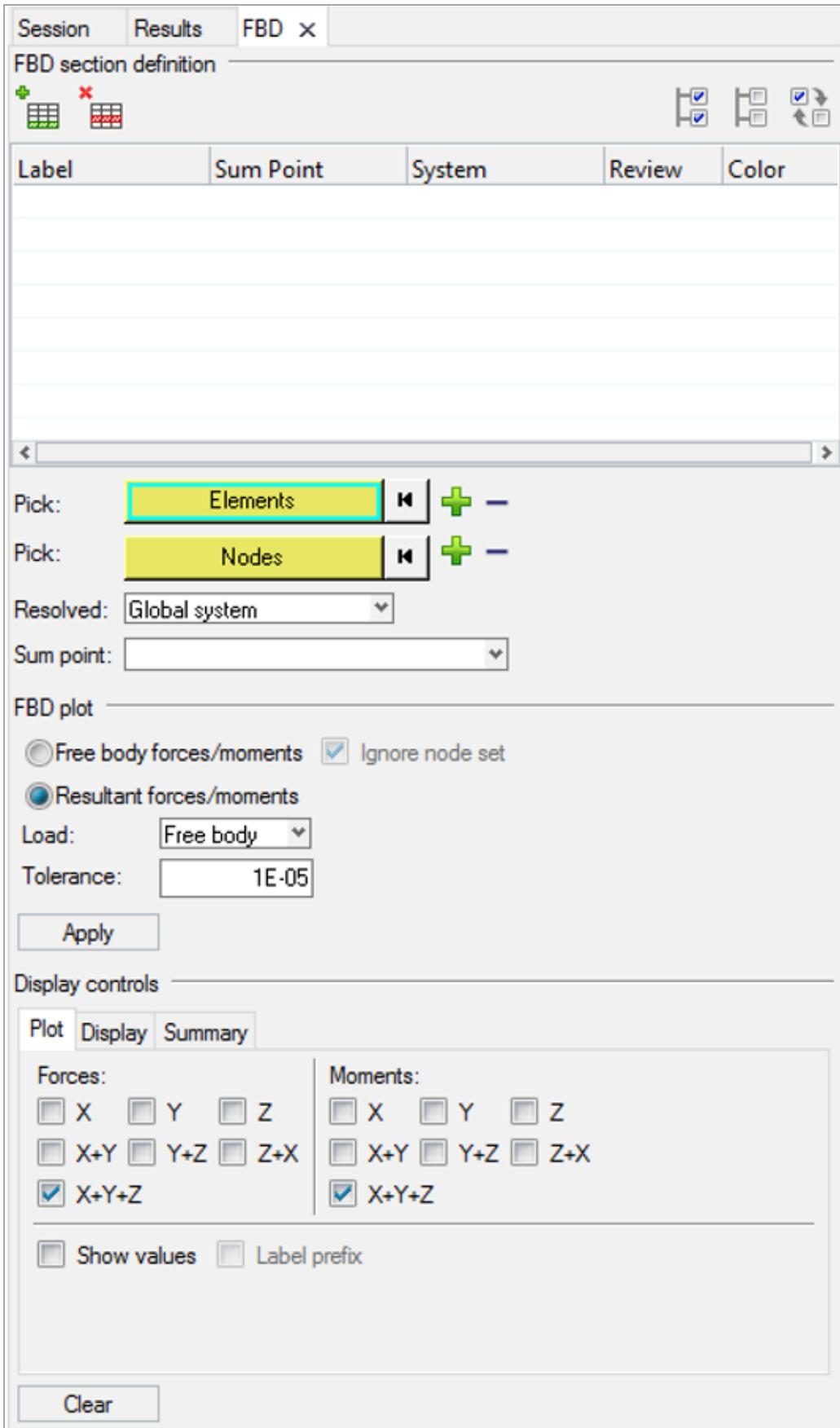
From the Results toolbar, click **FBD** .

Figure 2.

The FBD Forces utility is displayed in the tab area and is broken down into three major sections, each of which corresponds with the process order of using the tool:

FBD section   Element based sections are defined and modified in this first step of the

definition	process. You can define multiple sections, associate elements, associate a node group (optional for resultant summation), or define other attributes (like a summation point, a coordinate system, etc.).
FBD plot	Once the sections are defined, you can plot results (Freebody or Resultant Forces/Moments) in this middle step of the process.
Display controls	The presentation of the results/plots are controlled in this final step of the process. The various controls allow you to determine how the vectors are drawn in the modeling window, view the results in a summary table, or export/save the results to a `.csv` or `.fbd` file.

Create a Section Definition

Plot FBD Results

Create a Vector Plot

Change the Vector Display

View Results in a Summary Table

Export/Save Results

Manage Section Definitions

Delete a Section

Access the Context Menu

Clear an FBD Plot

FBD Solver Interfacing

See Also

Load Model

Entity Selection