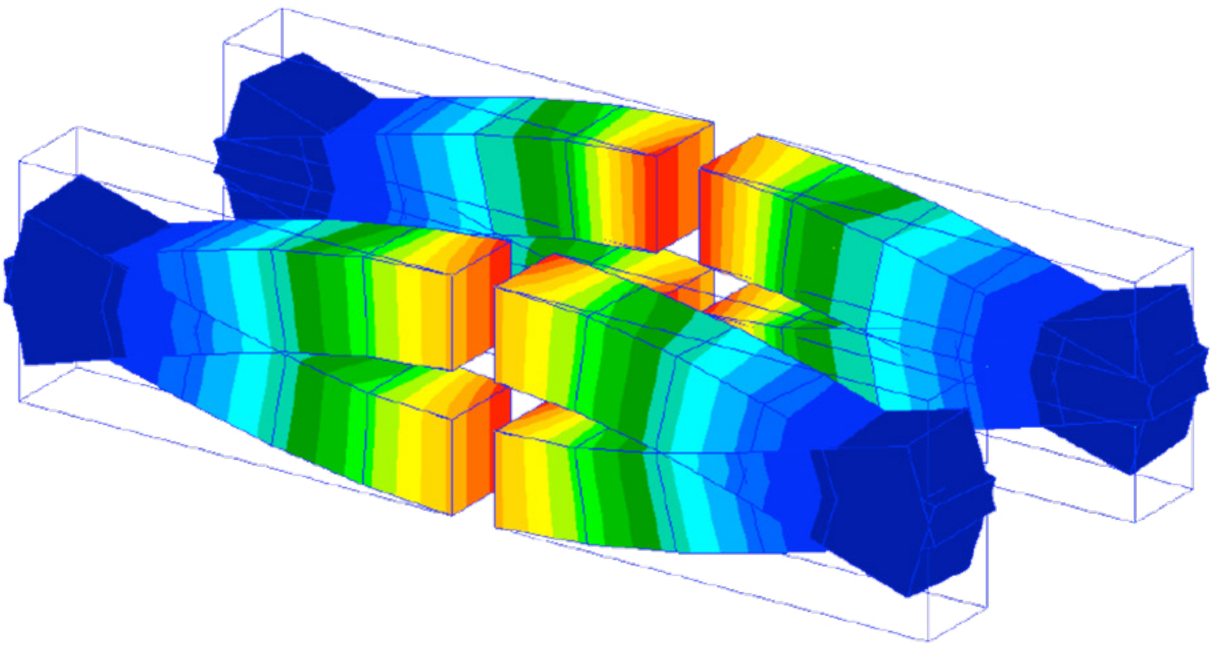
Patran® 2018



PART OF HEXAGON

MSC Software®

Welcome to Patran 2018!

MSC Software is pleased to announce the latest release of Patran 2018 with new enhancements that are designed to improve user productivity. Major areas of focus in this release include the following:

MSC Nastran – Patran Compatibility

MSC Nastran 2018.1 has been released to ensure improved compatibility with Patran 2018.0

This is a Shelf Replacement for the latest version of MSC Nastran (2018.0.1) and contains:

- Bug fixes related to forward compatibility with Patran
- Bug fixes related to HDF5 performance
- Bug fixes related to import/export of files

You can download this new product release at the [MSC Software Solutions Download Center] under MSC Nastran 2018 folder.

Enhanced Grid Point Force Display (GPFORCE)

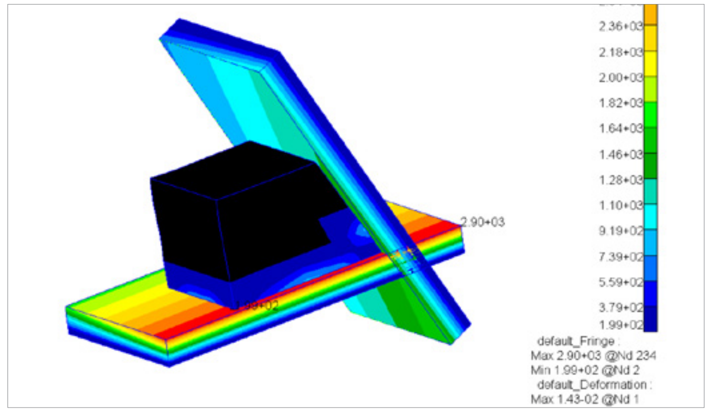
In MSC Nastran, GPFORCE output is used to provide the value of force at the grid points. This output clearly shows if the total force on a particular grid is balanced or not. It also provides users with the contribution and transfer of the forces in the structures. (for example, force contributions from MPCs, SPCs, Elements are provided to the user). Older versions of MSC Nastran and Patran, did not include contact and friction forces in the GPFORCE output. In some cases, this could lead into unbalanced GPFORCE output. Starting from version 2018, MSC Nastran supports both contact and friction forces in GPFORCE output. Similarly, Patran 2018 supports these new GPFORCE outputs for post processing via the HDF5 results file.

Contact and friction forces obtained from the GPFORCE output are supported for all types of contact models. This includes: permanent/general glued contact, touching contact, segment to segment and node to segment contact.

In Patran 2018, GPFORCE output data can be plotted in a variety of manners, such as: Fringe Plots, Vector Plots, and Freebody Plots or any other type of Patran data.

Available data in Patran include:

- Freebody Loads,
- Applied Loads
- Constraint (SPC) Forces
- MPC Forces, Contact Forces
- Friction Forces
- Internal Forces
- Summation of All Forces.



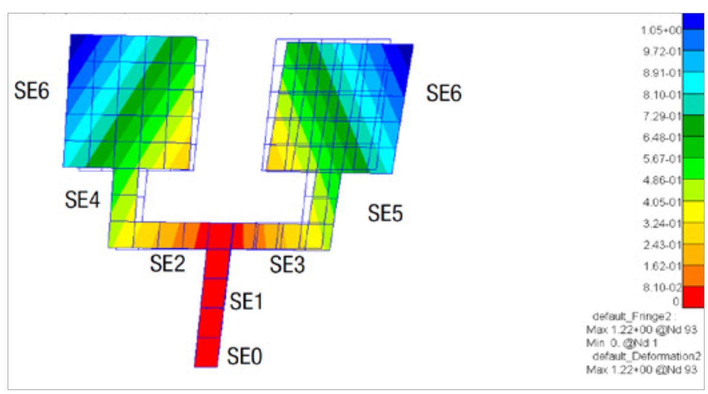
Fringe Plot of Grid Point Force Data; Contact Forces

Enhanced HDF5 Results Support

A new MSC Nastran database in Hierarchical Data Format (HDF5) was introduced in 2016. HDF5 is a database that stores and manages MSC Nastran input and output data in a hierarchical structure format. Although Patran has been supporting post processing of these result files, the 2018 release expands its capabilities to support additional output data through HDF5 files. Some of the supported outputs include:

- Grid Point Force Balance
- MSC Nastran Embedded Fatigue Results (supported for both time and frequency domains)
- Combined Superelement Results (for all supported solution sequences, 101-112, 200, and 400)

As an example, here we can see the HDF5 file containing results from multiple superelements can now be combined into a single result case for all superelements in the model. This provides better and easier plotting options for the user and is supported for both list and part superelement types.

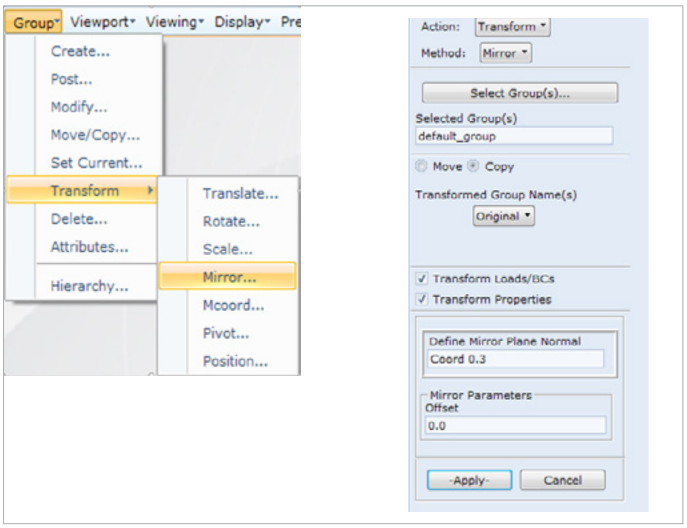


Post-processing of Combined Superelement (SE) Displacement Results with HDF5 Files

Enhanced Mirroring Options

The mirror option in Patran 2018 has been enhanced to provide easier options for users. This functionality is accessible through the Group/ Transform menu.

The Mirror option rigidly rotates all the entities in a group by 180 degrees across a mirror plane, thus it creates a mirror image. When LBCs and properties are mirrored, the finite element model results should be symmetrical.

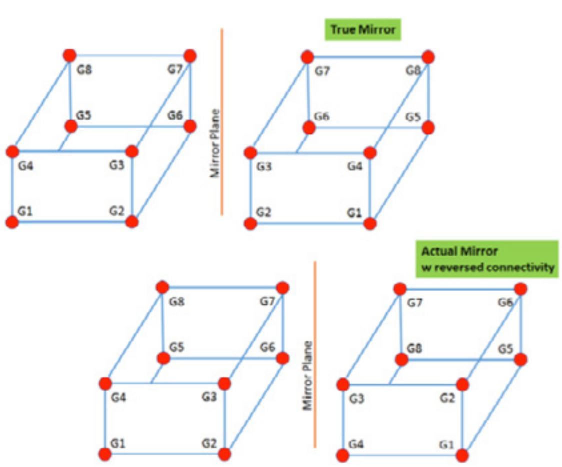


Important improvements include:

Connectivity Reversal of All Mirrored Elements

The nodal connectivity of all mirrored entities is automatically reversed relative to their original position. This provides better results for all element types, even after mirroring:

- The beam orientation vectors are properly defined,
- The shell element normal vectors are pointing in the mirrored direction,
- The mirrored solid elements are properly defined (no inside out/ negative volume solids).
- Geometric lines, surfaces, and solids are no exception and will be mirrored in a similar manner.



Transformation of Loads and Boundary Conditions (LBCs)

LBCs can also be transformed to produce a mirror image on the mirrored entities. This is especially important for directional LBCs, such as displacements, forces, moments and etc. Once mirrored, new coordinate systems are created for the directional (vector) LBCs as necessary.

When an LBC is defined as a discrete FEM field, the newly mirrored entities are added to the FEM field associated to the transformed vectors from the original entities.

Scalar LBCs (such as temperature) are duplicated as necessary for the mirrored elements or their application regions modified to contain the mirrored entities.

Pressure loads are not transformed as the connectivity reversal automatically keeps them pointing in the proper direction relative to the outward normals of the elements.

Transformation of Element Properties

When necessary, the element properties can be transformed for the mirrored elements as well. This includes all types of properties including those that are directional such as material orientations and offsets. New coordinate systems are created and referenced by the mirrored properties as needed.

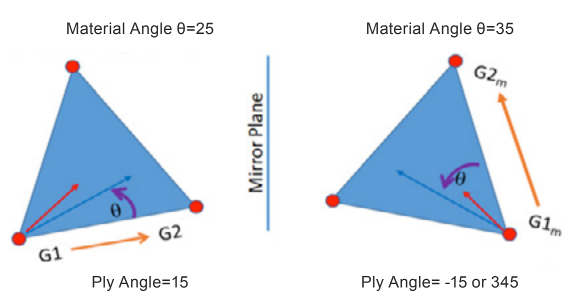
Beam properties with different values on each end are automatically swapped due to the connectivity reversal. Similarly, properties associated with element edges or solid faces can be swapped.

Properties defined by fields are evaluated at the original locations and then copied and associated to the mirrored elements or transformed as necessary in a similar manner.

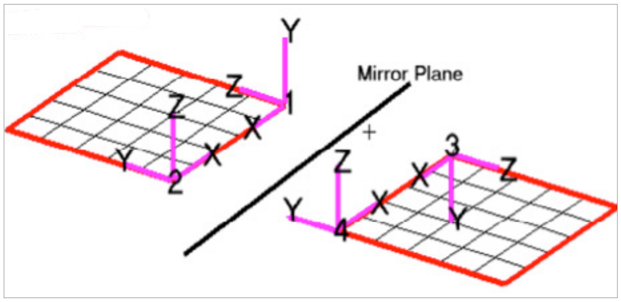
Transformation of Composite Ply Angles

Directional material properties (i.e. orthotropic) are affected by the Material Orientation of the elements. In MSC Nasran, users can define the shell element material orientation by specifying Theta (angle) or MCID (coordinate system). In addition to Theta and MCID methods, Patran allows the user to pick a Vector for defining the Element Material Orientation. For more details on each option please refer to the Reference Guides for both MSC Nastran and Patran.

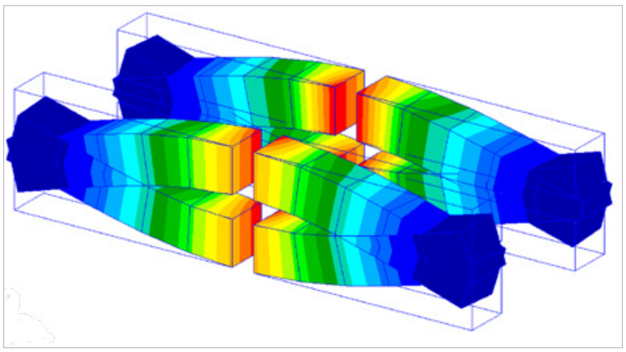
Mirroring of directional materials that are associated to composite laminates can have a significant effect on the quality of analysis results. When mirroring these elements, ply angles must be transformed properly in order to obtain a symmetric ply layup. Using Patran 2018, the laminate ply angles and element material orientation (regardless of the method used i.e. THETA, MCID or Vector) are automatically mirrored for all supported element types.



Theta Material Orientation Mirroring with Play Reversal



MCID Material Orientation Mirroring



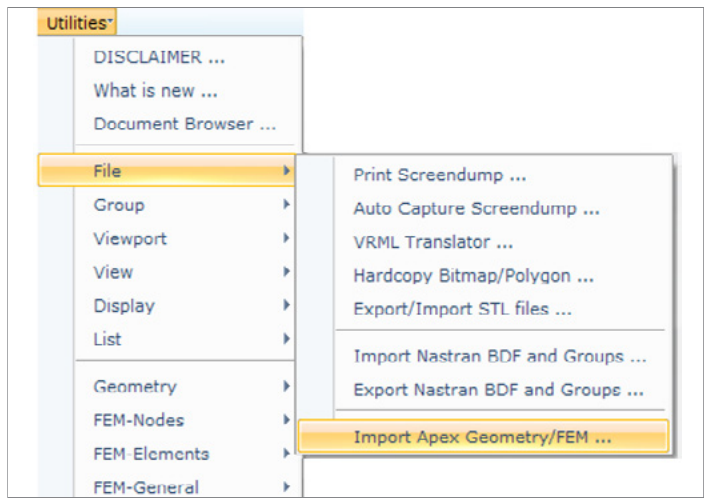
Mirrored Solid Composite Laminate across XY, YZ, and ZX Planes with Symmetric Results

Import of MSC Apex Models

Patran 2018 is designed to work with various CAD and FEA solvers to create analysis ready models for a wide range of simulations including linear statics, dynamics, nonlinear, Fatigue and etc.

For example, one can utilize the Direct Modeling and Meshing methods of MSC Apex to expedite repairing, editing and meshing the geometry and export it from MSC Apex to be used in Patran. The new utility in Patran 2018 can then be used to easily import the Geometry and FEM data into Patran. The users can easily continue the analysis in Patran without the need to worry about the associations between the elements, and the mesh.

This utility is designed to increase user productivity by expediting the process workflow between MSC Apex and Patran.



PATRAN*2018APR*RO