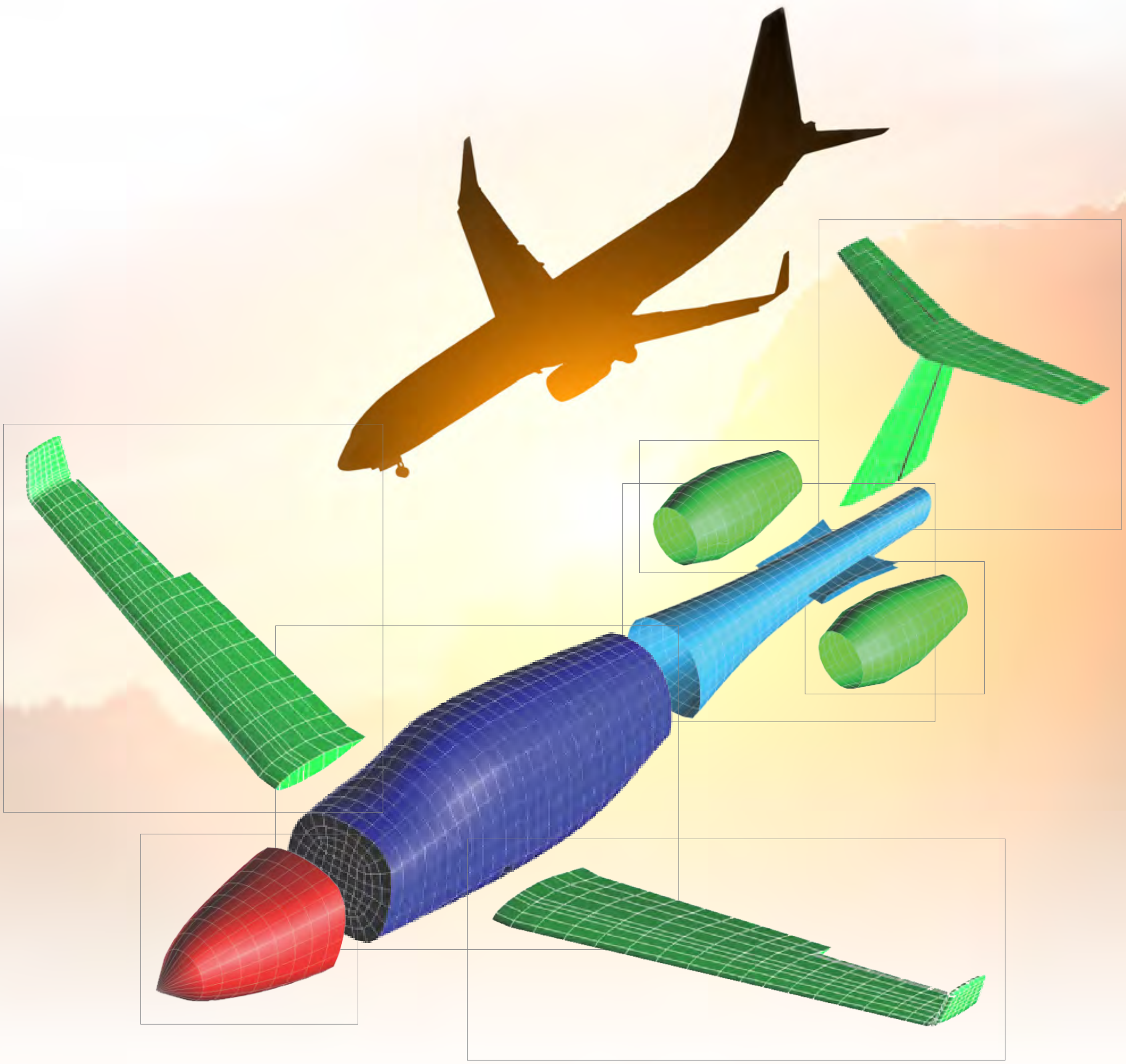
MSC Nastran™ 2018



Welcome to MSC Nastran 2018.0!

With more than 50 years of industry application, MSC Nastran has widely been used by various industries to solve linear static, dynamic, nonlinear, fatigue and many other engineering problems. MSC Nastran 2018.0 continues the evolution of better CAE design process by providing new material and element properties, large assembly modeling techniques, and improved contact analysis capabilities, all combined with more efficient solver methods.

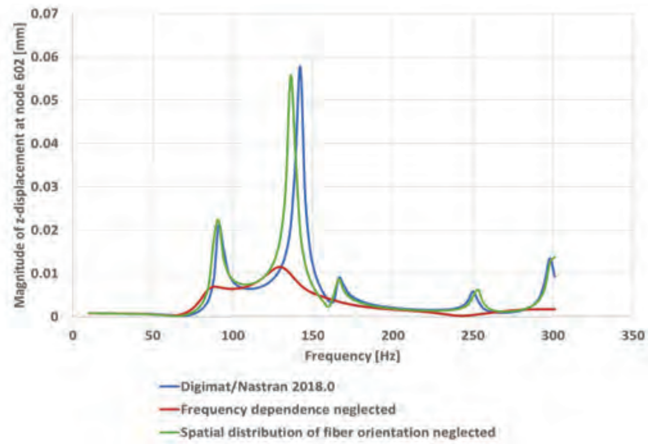
Frequency Dependent Material Properties

Besides safety, passenger comfort is another important area of focus for today's automotive and aerospace industries. This explains the high demand for usage of modern composite laminates for Noise, Vibration and Harshness (NVH) studies of cars and airplanes. Some examples include: laminated metals, laminated glass windscreens or layered fiber composites. In addition to noise abatement of vehicles, other applications include, design of spacecraft structures or integrally-bladed disks in turbomachinery. These materials exhibit frequency dependent damping and stiffness properties, meaning their damping values can change per frequency. Accurate representation of material properties can directly impact the analysis quality and subsequent results (i.e. structural and acoustic responses). Using the new MAT(i)F entries of MSC Nastran 2018.0, engineers can accurately account for the frequency dependent behavior of materials in both Direct and Model Frequency Response solution sequences (SOL 108 and SOL 111).

Figure 1: Laminated metals (where two thin sheets of steel or aluminum are bonded together by a thin frequency-dependent damping layer).

Spatially Dependent Element Properties

Modern composite materials that are lightweight and have high stiffness to weight ratio are becoming more prevalent in both automotive and aerospace industries. They are often molded (short-fiber) or draped into complex structural shapes that are used for many high-end products. Proper implementation of these materials often requires users to change the thickness or orientation of certain plies for a specific group of elements in the model. MSC Nastran 2018.0 offers a general interface to easily define spatial dependent properties, eliminating the need to have a separate property (i.e. PCOMP entry) for each element in the model.

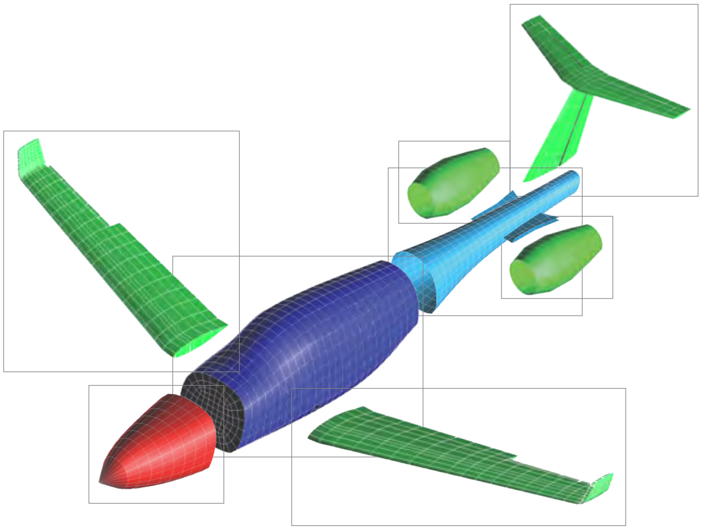


Accounting for frequency-dependence and spatial distribution of fiber orientation has significant impact on analysis results.

Easier Assembly Modeling with Modules

Complex structures consist of various parts that represent a section of the overall assembly. Handling and connecting these parts (mostly coming from different sources) requires cumbersome modeling techniques. To overcome this issue, MSC Nastran 2018.0 provides new features that make substructure assembly management much easier for large scale models.

- Use Modules to easily create, combine, and manage multilevel assemblies.
- A module is a standalone section of the MSC Nastran input file that represents a component of a full assembly (i.e. a wheel or a fender is a module that is part of the full automobile assembly).
- Modules provide an independent process workflow, meaning that analysts can work on each module independently.
- Modules are similar to part superelement without the reduction
- Modules offer a complete range of flexibility when it comes to defining inter module connections. Manual connections range from flexible fasteners and bolts to rigid elements like RBEs.
- Automatic and semi-automatic module connections are available to the user. In the AUTO case, MSC Nastran automatically connects the grids from a Module to any coincident grids in other Modules. There is no need to be concerned with coordinate systems on these coincident points, as the solver accounts for different output coordinate systems.
- Modules may be specified along with advanced analysis features such as HPC methods, fatigue, contacts and etc.
- Results are presented in a module-by-module basis which makes it much easier for post processing purposes.

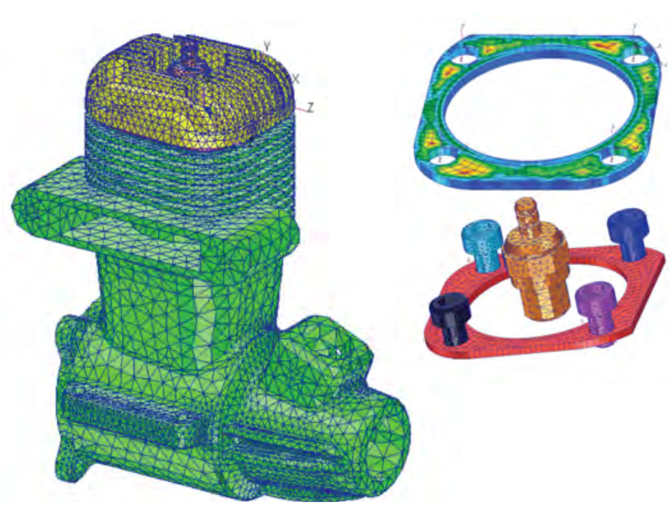


Modules representing different components of the model

Automatic Generated Contact

Designed to increase user productivity, MSC Nastran 2018.0 provides new techniques that simplify creation of multilevel assemblies. The new Automatic Contact Generation (ACG) feature allows you to easily create contact bodies and establish the contact relationship between them. ACG is designed for quick model setup of complex structures and helps you study the interaction of parts that come into contact under applied loads.

- ACG is available for both Glued and Touching contacts.
- Users have an option to output the generated contact bodies, contact pairs and contact parameters in a separate file. The data on the file can be read into a pre/post processor for either model verification or result evaluation.
- With ACG, one can save time by getting a reasonably good set of contact bodies and pairs without going to a UI. This reduces reliance on a pre-processor.
- Parameters such as ESet, PropSet, SeedESet and SeedGSet give user more control on elements picked to define contact regions.



Accounting for frequency-dependence and spatial distribution of fiber orientation has significant impact on analysis results

Multi Mass Configuration

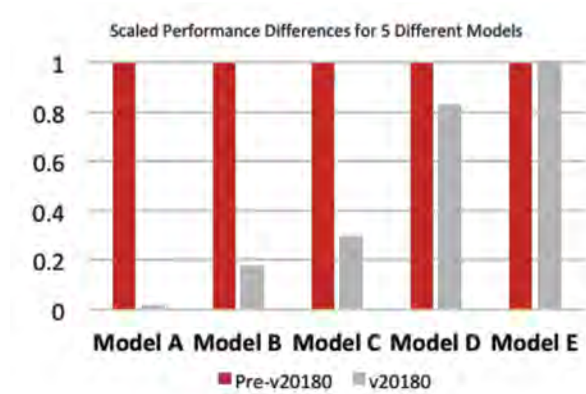
Complex aerospace or automotive vehicles require users to account for multiple mass configurations such as payloads, fuel conditions, passenger count, bed loads and other environmental conditions. This allows engineers to conduct durability studies a single run - a major efficiency workflow improvement. Before this new release of MSC Nastran, analysts had to use workarounds such as subcase dependent MPCs or multiple runs to simulate these mass cases.

- MMC is supported in most MSC Nastran solutions, including linear statics and dynamics, nonlinear static and transient, aeroelastic analysis and optimization (supported with invariant mass increments).
- User can easily add various mass configurations to the base mass and run a single analysis with all mass cases.
- The mass cases are subcase selectable to - A particular mass case can easily be included or removed from the analysis similar to selecting SPCs and MPCs in a subcase.

High Performance Computing

Analysis models that are very large in size, require an extended period of time to solve. MSC Nastran features new High Performance Computing (HPC) enhancements that provide optimal solutions and enable engineers to solve large problems much faster.

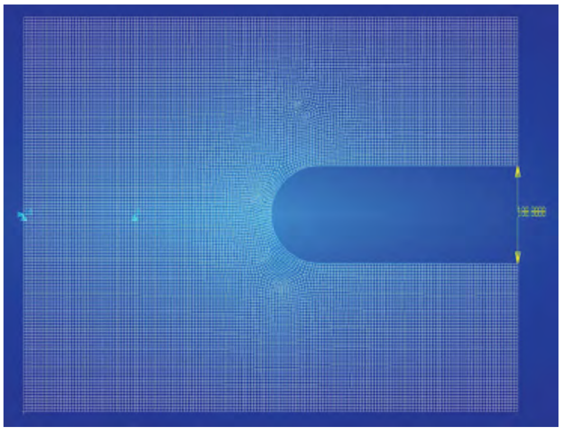
- Most of matrix multiplication operations in MCS Nastran are handled by MPYAD module (stands for multiply and add). In this release, MSC Nastran has a completely new version of MPYAD that is designed to improve matrix operations during the solving process which can result in up to 5 times faster performance for many models.
- Parallelization methods of MSC Nastran 2018.0 have been improved for ACMS method. The new method is designed to make better use of shared memory parallel (SMP) processing. This means more threads are busy working on the solution throughout the computation process which leads to improved performance and better solver speed.
- MSC Nastran 2018.0 is featured with new Automatic Solver Selection methods that are designed to automatically select the optimal solver and parallelization methods for each model.
  1. Selects the best matrix or modal solver based on the analysis specifications (i.e. Casi solver is selected for solid dominated models in SOL 101 and 400).
  2. In SOL 101 and 400, machine learning techniques have been added to predict the memory required by the Pardiso Solver and actually improve on those predictions over time.
  3. Selects the best DMP/SMP parallelization method.



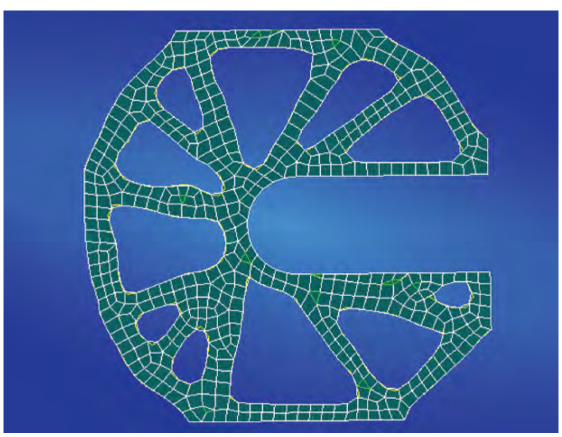
Optimized calculation methods can result in up to 5 times performance gain for some models.

Topology Optimization

In previous versions, Stress constraints for solid element topology optimization were introduced. For the 2018 release, stress constraints are extended to support shell element topology and topometry optimization. Generally the goal of most toplogy optimization analysis models is to maximize structural stiffness while confining to the existing constraints (mass targets). Since specifying a meaningful fractional mass limit is not a very easy task, most applications often recommend to create multiple runs with different mass targets. Availability of stress constraints in MSC Nastran allows users to specify a natural way for finding the optimal design without including a mass target. Inclusion of stress constraints provides a more practical design proposals that considers global stress levels in the model.



In this example, the dimensions of the clip are 100 horizontally and 80 vertically.



Minimize FRMASS s.t. stress <=100.0MPA

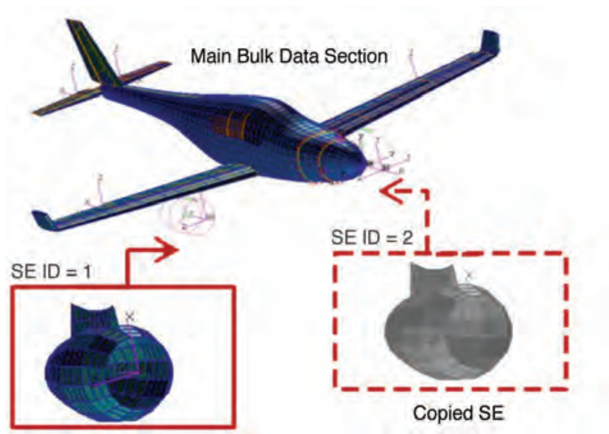
Rotor Dynamics

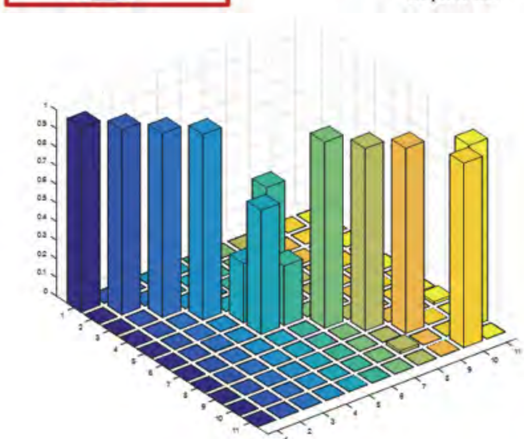
In this release, several improvements have been added to improve the efficiency of Rotordynamic analysis and provide additional output.

One of the key steps for most of the Rotordynamic analyses is generation of the Campbell diagram which demonstrates how rotor frequencies are varied with respect to rotor speed. During the analysis, the order of rotor modes may switch over the range of rotor speeds. Therefore, it is important to have a mode tracking method to identify and track a particular mode for the whole speed range. Earlier versions of MSC Nastran did not provide any mode tracking capability for Rotordynamics and users had to perform this analysis outside MSC Nastran. In the new MSC Nastran 2018.0 release, mode tracking capability is added inside the MSC Nastran code and it features two methods: Numerical-based and Eigen-vector based. These new mode tracking algorithms are avilable in Direct Complex Eigenvalue Analysis (SOL 107) and Modal Complex Eigenvalue Analysis (SOL 110) solutions.

Additionally MSC Nastran 2018.0 supports modeling of rotors in single/ multi-level part superelement assemblies. This feature can significantly improve the efficiency of Rotordynamic analysis.

Other enhancements include identification of whirl mode type for complex Eigenvalue analysis solutions. Earlier, this process was done using by looking at the mode's whirl behavior in a post-processing software like Patran or by looking at the variation of a particular mode in a Campbell Diagram. In V2018, this feature is only available for 1D rotors defined using ROTORG bulk data entry.





MSC-N*2018NOV*RO