MSC Apex[®] Structures Computational Parts Based Structural Analysis

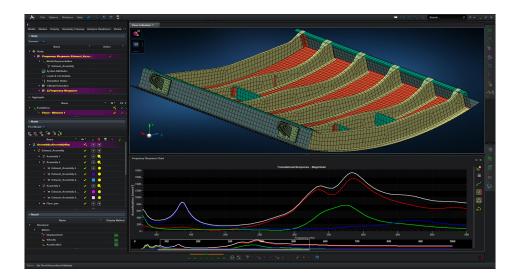
Overview

MSC Apex Structures is an add-on product which expands MSC Apex Modeler functionality with capabilities for linear structural analysis.

MSC Apex structures packages a user interface for scenario definition and results post-processing, as well as integrated solver methods. This solution is unique in that it combines computational parts and assemblies technology with a generative framework, which enables interactive and incremental analysis.

The integration of the user interface with solver methods gives the user a unique ability to interactively and incrementally validate that FEM models are solver ready. At the user's demand, a series of solver checks can be run against individual parts and assemblies and the model diagnostics are reported in the Analysis Readiness panel. This Incremental Validation is a radical departure from the very time consuming traditional approach where pre/post processor and solver are separate.

In addition, a frequency response analysis type and a specialized results exploration toolset is available to aid engineers improve the vibration behavior of structures. The integrated toolset of MSC Apex enables analysts to experiment with mode contributions and develop design solutions to mitigate and control structural vibrations, all without committing to excessive modeling changes and re-analysis.



Capabilities

Generative Framework

 Geometry, Mesh, Material, Property and Behaviors, Glue, Load and Boundary Conditions, Scenarios and Results automatically update with changes to the model

Incremental Validation

- Context specific (Part, Sub-assembly, Assembly)
- Regenerative Analysis Readiness for mesh, materials, properties, LBCs, interactions, and simulation settings to guide the user on quickly achieving run ready models

Incremental Solve

- Computational Parts and Assemblies Linear Structural Analysis

• Linear Structural Analysis

- Linear Statics
- Normal Modes
- Frequency Response
- Specify a multi-step Frequency Response Analysis: 1) Pre Stiffening (optional), 2) Normal Modes, 3)
 Frequency Response Analysis

Results View

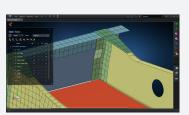
- Use a hot spot tool to identify critical displacements and stresses
- Animate deformed shapes
- View and interactively switch between multiple normal modes via modes navigator
- Use a Results Manager to view analysis results by study, part, assembly or result type
- Transform results to Cartesian, cylindrical or spherical coordinate systems
- View fringe color plots of displacements, stresses, strains, etc.
- Vector plots of displacements, applied loads, constraint reactions, and more
- Create Sensors and monitor responses at specific points such a displacements and stresses
- Display results in XY plots
- Study Manager
 - Manage multiple scenarios (model representations, output requests, analysis type)

Structural Analysis Workflow



Set model and analysis context

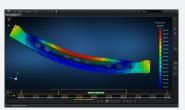
Define the analysis type and a subset of parts and assemblies to be the context of evaluation



2

Validate models prior to analysis

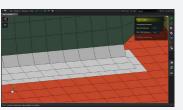
Use the integrated analysis readiness tool to validate the context has valid model representations for the chosen analysis type





Join dissimilar meshes rapidly

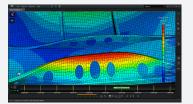
Reduce the need to align nodes across mesh parts using mesh independent glue technology





Generate and visualize results for Linear Statics

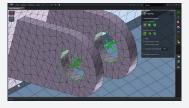
Define a linear static or normal modes scenario and execute the integrated solver methods to generate results interactively



4

Make generative changes

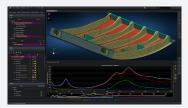
Track the status and manage the update of downstream updates whose parent has been modified



6	

Generate and visualize results for Frequency Response

Perform frequency response analysis & use a results exploration toolset to develop solutions to mitigate and control structural vibrations.



Productivity Gains Linear Static Analysis For this landing gear door 1400 Results assembly, Computational 1200 Parts technology was used S to perform an incremental Recover analysis. After modifying ■ Solve Assemble one part of the assembly, an Generate incremental or subsequent analysis completed 2.5x faster than its first solve. Model

Corporate

MSC Software Corporation 4675 MacArthur Court Suite 900 Newport Beach, CA 92660 Telephone 714.540.8900 www.mscsoftware.com

Europe, Middle East, Africa

MSC Software GmbH Am Moosfeld 13 81829 Munich, Germany Telephone 49.89.21093224 Ext.4950 Japan MSC Software LTD. Shinjuku First West 8F 23-7 Nishi Shinjuku 1-Chome, Shinjuku-Ku Tokyo, Japan 160-0023 Telephone 81.3.6911.1200 Asia-Pacific MSC Software (S) Pte. Ltd. 100 Beach Road #16-05 Shaw Towers Singapore 189702 Telephone 65.6272.0082

MSC Software

First Solve

The MSC Software corporate logo, MSC, and the names of the MSC Software products and services referenced herein are trademarks or registered trademarks of the MSC.Software Corporation in the United States and/or other countries. All other trademarks belong to their respective owners. © 2016 MSC.Software Corporation. All rights reserved

Incremental Solve