Overview

The MSC Apex Frequency Response capability features specialized methods meant to aid engineers to 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
  - Context specific (Part, Sub-assembly, Assembly)
  - Regenerative Analysis Readiness for mesh, materials, properties, LBCs, interactions, and simulation settings

- **Incremental Validation**
  - Analysis readiness for mesh, materials, properties, LBCs, interactions, and simulation settings
  - Context specific (Part, Sub-assembly, Assembly)

- **Incremental Solve**
  - Computational Parts and Assemblies
  - Linear Structural Analysis
  - Linear Statics, Normal Modes, Linear Buckling, and Frequency Response Analysis
  - 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)
**MSC Apex Frequency Response Workflow**

1. **View frequency response results**
   - Review plots of displacements, velocities, and accelerations vs frequency and actively switch results across different channels or sources of excitation.

2. **Identify highest contributing modes**
   - Use a histogram to identify highest contributing modes.

3. **Experiment with mode contributions**
   - Adjust modal contributions, frequency shifts and damping and immediately view impact on frequency response.

4. **Make design changes**
   - Use Direct Modeling and Meshing technology to rapidly make changes to part geometry and meshes that will lead to different structural vibration behavior.

5. **Reconfigure assembly layouts**
   - Modify assembly configurations to influence vibration behavior, while the MSC Apex Generative Framework preserves and updates previously created loads, constraints, connections, meshes, etc.

6. **Review validated responses**
   - Perform a subsequent frequency response analysis to validate model change has led to the desired vibratory affect.