

CoCreate® Finite Element Analysis

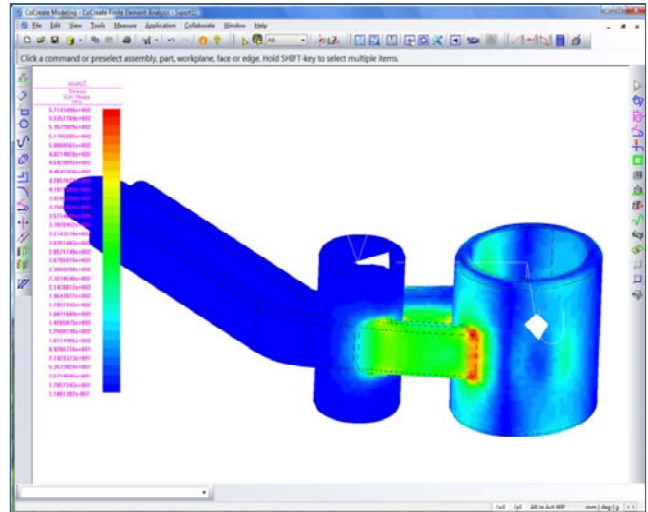
Simulate real-world stresses on your design

CoCreate Finite Element Analysis offers an extensive set of finite element analysis capabilities for engineers and designers.

Based on the MSC Nastran® and Patran® solutions, CoCreate Finite Element Analysis gives you all the power you need to simulate real-life mechanical and thermal stresses on products directly within the CoCreate Modeling 3D CAD system.

Faster identification and resolution of design issues helps companies transition from a stage-gate process for design and analysis to a more efficient product development process where digital simulation occurs as part of the daily design activity.

Easily perform a variety of FEA studies for structural, buckling, thermal, and frequency analysis using various working load and boundary conditions that you apply to your product design.



Run CoCreate Finite Element Analysis simulations on your virtual product model

Key Benefits

- Achieve faster realization of optimal designs, avoid both failure-prone and over-engineered components, and reduce physical prototyping costs
- Identify areas within a product that will be prone to failure. Issues can be resolved early in the design process, increasing design quality and saving costs and time
- Conduct extensive trade-off studies upfront in the development process and yield downstream benefits including increased design quality, reduced time to market, reduced cost of goods sold and reduced warranty exposure

Features and Specifications

Complete and Fully Integrated Solution

- Perform linear and static analyses
- Execute all analyses within the CoCreate Modeling environment
- Set up and store loads and boundary conditions. Plus, assign material types directly with the part or assembly data
- Store multiple studies of the same part or assembly data
- Leverage automatic meshing and solving for parts and assemblies
- Full-color visualization of part showing clearly stress, strain, displacement, etc.
- Store results with the part
- Animate results and document using HTML

Analysis Cases

- Analyze stress levels, displacements, and resonant frequencies of designs. Supported cases include:
 - Linear static structural analysis
 - Linear buckling
 - Normal modes
- Analyze thermal cases:
 - Steady state thermal
 - Solving for temperature and flux

Loads and Boundary Conditions

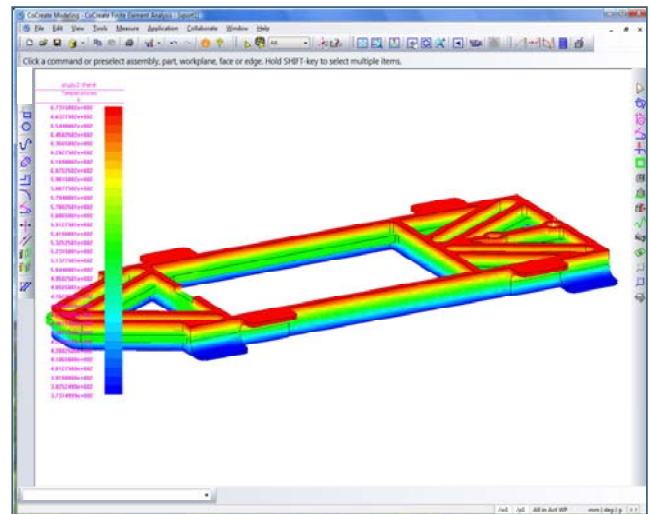
- Assign LBCs, directly to part or assembly, including:
 - Vertex, edges, and face loads
 - Spin, gravity, acceleration, and part temperature
 - Face pressure loads
 - Translational constraints and enforced displacements (XYZ) for vertex, edge, or face
 - Heat flow, heat generation, free convection, fixed temperature

Materials

- Leverage the standard database which includes over 900 commonly used materials such as multiple types of steel, aluminum, and plastics
- Add more materials and material parameters

Meshing

- Generate volumetric meshes automatically using technology provided by MSC Software Corporation
 - Tetrahedral linear or quadratic order adaptive h-element technology
 - P-element technology
- Automatically create shell-elements tailored for sheet metal part analysis
- Generate surface mesh with triangular or quadratic h-elements
- Export mesh with or without LBC data as a PATRAN®-neutral file
- Apply mesh conditions to the part or assembly and control local mesh density



Easily detect weaknesses in your design before using expensive tests with physical prototypes

Solving

- Solve cases with unlimited nodes and unlimited mesh size. Solving based on the latest h- and p-elements technology from industry leader MSC Software Corporation

System Requirements

CoCreate Finite Element Analysis supported operating systems:

- Windows Vista™ 32-bit and 64-bit Editions of Ultimate, Enterprise, and Business
- Windows® XP™ Professional 32-bit and 64-bit Editions

For more information, visit www.ptc.com/products/cocreate