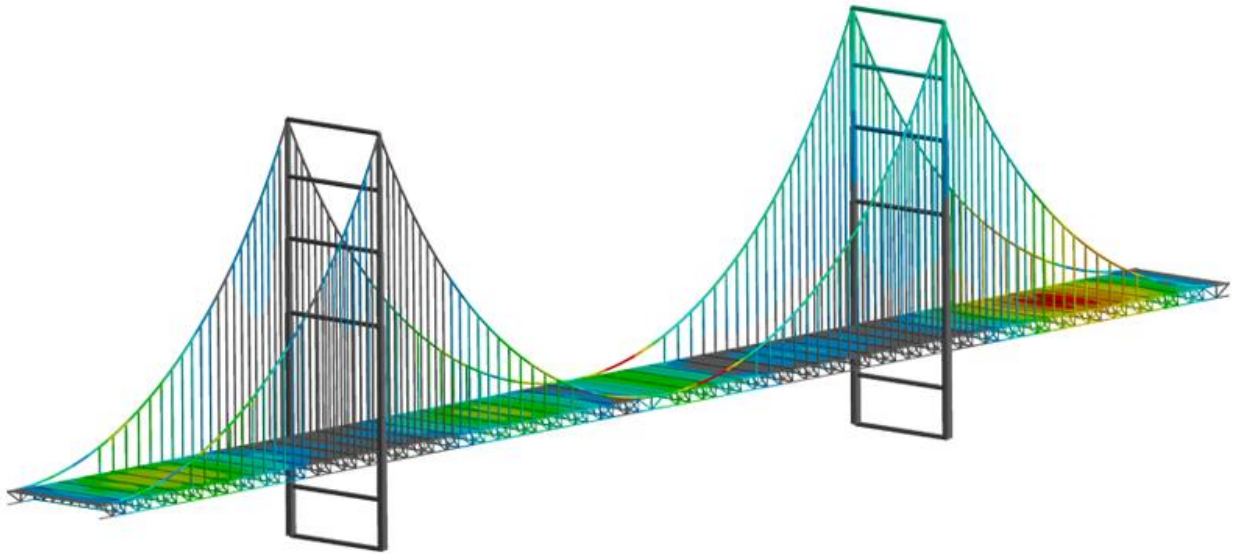


ANSYS STRUCTURES Update in 2020 R1

ANSYS 2020 R1 empowers ANSYS Mechanical users to go further than ever before with enhancements to improve the handling of complex, highly nonlinear and massively large models. Enhancements include:

Additional functionality directly in Mechanical for streamlined workflows, including cross sectional assignments for line bodies, post-processing of reinforcements and easy drag-and-drop external models.



Improved topology optimization validation workflows so you can now go directly from a tessellated optimal shape to meshing and validation of the final design

Full migration of ANSYS AQWA, a hydrodynamic analysis tool, into Mechanical, including new technology such as the ability to transfer generated loads to other analysis systems

Enhanced integration of LS-DYNA into the ANSYS Mechanical interface following our recent acquisition of LSTC

Quicker solve times in ANSYS Sherlock, which now uses ANSYS MAPDL as its default FEA engine, along with consolidation of application settings and the ability to configure multiple part libraries.

ANSYS structural analysis software enables you to solve complex structural engineering problems and make better, faster design decisions. With the finite element analysis (FEA) solvers available in the suite, you can customize and automate solutions for your structural mechanics problems and parameterize them to analyze multiple design scenarios. You can also connect easily to other physics analysis tools for even greater fidelity. ANSYS structural analysis software is used across industries to help engineers optimize their product designs and reduce the costs of physical testing.

Structural analysis for all experience levels

From designers and occasional users looking for quick, easy and accurate results, to experts looking to model complex materials, large assemblies and nonlinear behavior, ANSYS has you covered. The intuitive interface of ANSYS Mechanical enables engineers of all levels to get answers fast and with confidence. Solve on your local machine or [on the cloud](#). You can even run hundreds of design points using [ANSYS distributed compute services \(DCS\)](#).

Reliable, high-quality, automated meshing

Mechanical has intelligent meshing technology so you can rapidly obtain optimal meshing on every model. Easily added controls enable you to fine-tune the mesh as needed.

Advanced capabilities

Simulation of complex materials and material behavior can be achieved using the built-in models, user-defined material models or Material Designer in Mechanical to create representative volume elements (RVEs). Models with very large deformation need not be simplified by switching to explicit solvers unnecessarily. The nonlinear adaptivity (NLAD) capability automatically handles challenging simulations by remeshing the solution as it progresses.

Splitting morphing adaptive remeshing technique (SMART) fracture technology has been added to Mechanical for modeling crack growth in structures where fracture is a concern, so complex meshing processes and guesswork can be eliminated.

ANSYS Motion is a next-generation engineering solution based on flexible multibody dynamics within the Mechanical interface. It enables fast and accurate analysis of rigid and flexible bodies within a single solver. A series of toolkits provides extensions to ANSYS Motion’s functionality so you can investigate the behavior of a vehicle’s powertrain design, including gears and bearings, and systems incorporating chains, belts and continuous tracks.

Additive manufacturing

ANSYS provides a comprehensive, scalable software solution which minimizes the risk of your additive manufacturing processes and ensures high-quality, certifiable parts. With ANSYS’ additive solutions, you can go from designing for AM (DfAM) all the way through part qualification and certification with the capability to do build-file preparation, metal AM build process simulation and material analysis at a microstructure level.

Complete structural analysis solution

A complete range of analysis tools is available to analyze single load cases, vibration or transient analysis; you can also examine linear and nonlinear behavior of materials, joints and geometry. Advanced solver technology with ANSYS Autodyn and ANSYS LS-DYNA enables you to carry out drop, impact and explosion simulations. ANSYS AQWA, along with the offshore simulation capabilities in Mechanical, provides industry-specific capabilities for engineers designing marine environments.

Applications

STRENGTH ANALYSIS

VIBRATION

THERMAL ANALYSIS

DURABILITY

RIGID BODY DYNAMICS

HYDRODYNAMICS

COMPOSITES

IMPACT

OPTIMIZATION

HPC FOR FEA

TOPOLOGY OPTIMIZATION

ADDITIVE MANUFACTURING







