EnginSoft is a premier consulting firm in the field of Simulation Based Engineering Science (SBES) with a global presence. It was founded in 1984, but its founder and initial employees had been working in SBES since the mid '70s. Throughout its long history it has been at the forefront of technological innovation and remains a catalyst for change in the way SBES and CAE technologies in general are applied to solve even the most complex industrial problems with a high degree of reliability.

Today, EnginSoft is comprised of groups of highly qualified engineers, with expertise in a variety of engineering simulation technologies including FEM Analysis and CFD, working in synergic companies across the globe. We are present in Italy, France, Germany, the UK, Turkey and the U.S.A. and have a close partnership with synergetic companies located in Greece, Spain, Israel, Portugal, Brazil, Japan and the U.S.A.

EnginSoft works across a broad range of industries that include the automotive, aerospace, defense, energy, civil engineering, consumer goods and biomechanics industries to help them get the most out of existing engineering simulation technologies.





ITALY info@enginsoft.com

FRANCE info.fr@enginsoft. com

GERMANY info.de@enginsoft.com

UNITED KINGDOM info.uk@enginsoft.com

TURKEY info.tr@enginsoft.com

USA info@enginsoftusa.com



www.enginsoft.com | info@enginsoft.com

ENGINSOFT

DATA SHEET

GENESIS Topology for ANSYS Mechanical



- and Random VibrationsA
- casting, extrusion, etc.

C: GENESIS	
Topology Density Isosurface	
Expression: RES311 (Universiged)	
Time: 14	
2/19/2814-4:18 PM	
c 🗖	1 Max
	1.0000
	ILTTTTS
	1.00057
	1.55556
	1.4444
	1 3333
	1 20222
	1.11111
	0 Min



GENESIS® Topology for ANSYS Mechanical (GTAM) is an integrated extension that adds topology optimization to the ANSYS environment.

Designers benefit by automatically generating innovative designs in a reliable, robust, and easy-touse interface.





C6.R Tail Frame Assembly Courtesy of Pratt & Miller Engineering GTAM is a product



GENESIS Topology for ANSYS Mechanical

✓ Support multiple loading conditions and different analysis systems including Static Structural, Modal, Linear Buckling, Harmonic

✓ Several built-in responses for objectives and constraints such as strain energies, frequencies, mass, displacement, etc.

✓ Several built-in manufacturing constraints for the designable region such as symmetries,

