

BladePro™ - ANSYS® Workbench Interface



Utilizing Workbench is a simple process that provides total control over the analyses.

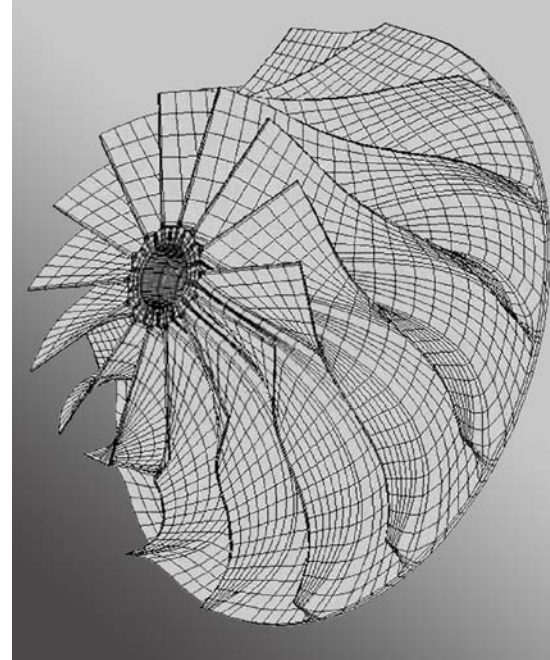
By utilizing the CAD input, ease of use, and meshing capabilities of ANSYS Workbench BladePro has the capability to work with CAD models.

BladePro has traditionally offered geometry input via a Graphical User Interface (GUI) or from text files. While this works well for parametric studies and digitized data many users' want a direct link to ANSYS Workbench and a way to import solid models from their CAD System.

Users can now take their model, meshed in Workbench, directly into BladePro. BladePro will add the appropriate boundary conditions, control the analyses and provide post-processing features common to turbomachinery analysis.

CAD Model

Step 1: The user creates a model of the turbine blade or impeller in CAD.



ANSYS Workbench - User Controlled Mesh

Step 2: The model is brought into ANSYS Workbench and meshed.

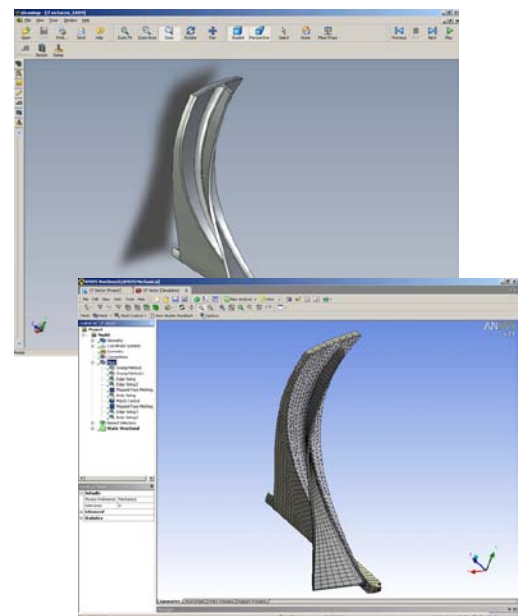
Boundary Conditions - User Controlled Boundary Conditions

Step 3: The Workbench model is brought into BladePro and boundary conditions are added by the user via a user-friendly GUI.

Analyses and Post-Processing - User Controlled

Step 4: Steady stress, modal, harmonic response, and fatigue life analyses are performed.

CAD Model



ANSYS Workbench
user controlled Mesh

impact-tek.com

© 2010, Impact Technologies, LLC

**Headquarters
New York Office**
200 Canal View Boulevard
Rochester, NY 14623
Phone: 585.424.1990
Fax: 585.424.1177

Pennsylvania Office
270 Walker Drive, Suite 200W
State College, PA 16801
Phone: 814.867.5122
Fax: 814.867.7550

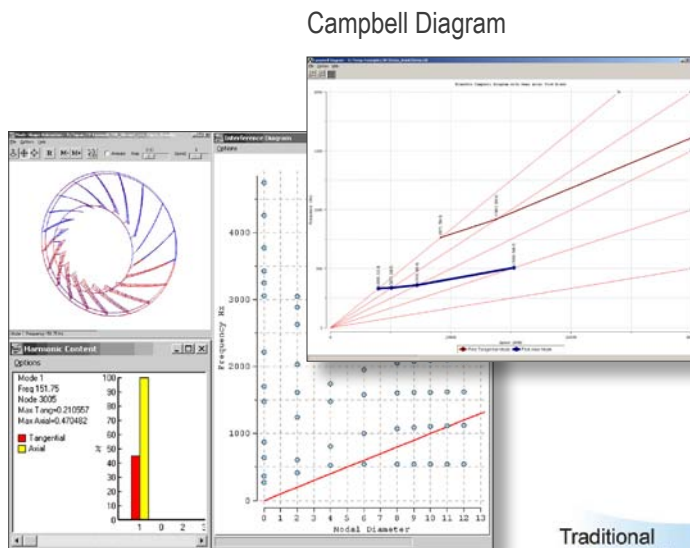
Georgia Office
75 Fifth Street NW, Suite 312
Atlanta, GA 30308
Phone: 404.526.6188
Fax: 404.526.6189

Impact engineers have applied their years of experience to ensure that BladePro conducts the *most cost-effective analysis*.

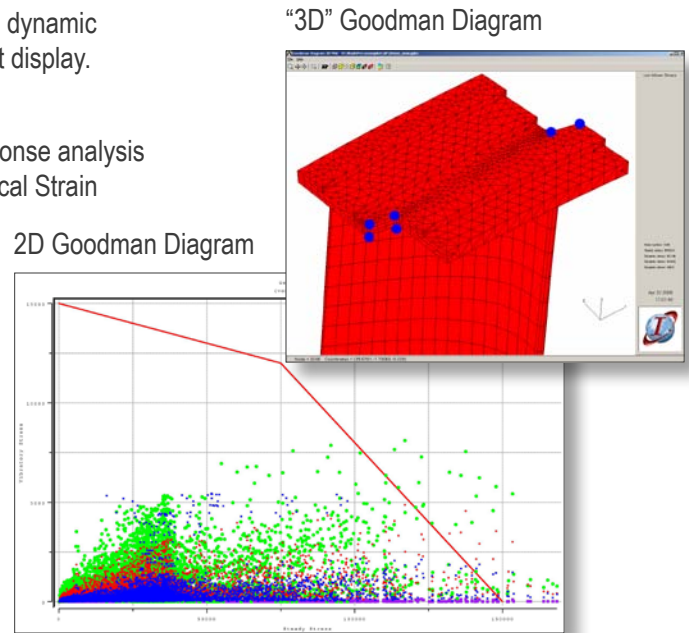
Dedicated Post-Processing

BladePro-AF and -CF provide automated tools for taking the results of a modal analysis and creating Campbell and Interference Diagrams, steady and dynamic stress displays, animated mode shape displays and a harmonic content display.

Combining the results from a steady stress analysis and harmonic response analysis directly leads to the creation of 2D and 3D Goodman diagrams. The Local Strain Method is used for more detailed fatigue analysis.

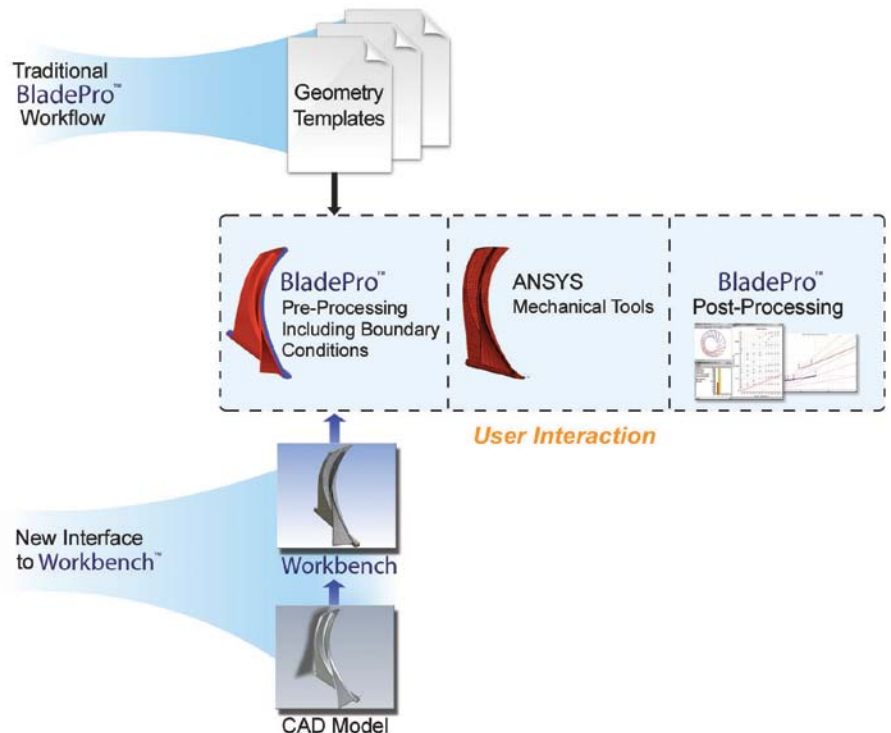


Animated Mode Shapes, Harmonic Content Display, Interference Diagram



2D Goodman Diagram

"3D" Goodman Diagram



ANSYS® is a registered trademark of ANSYS, Inc.