

# BladePro™ - ANSYS® Workbench Interface



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

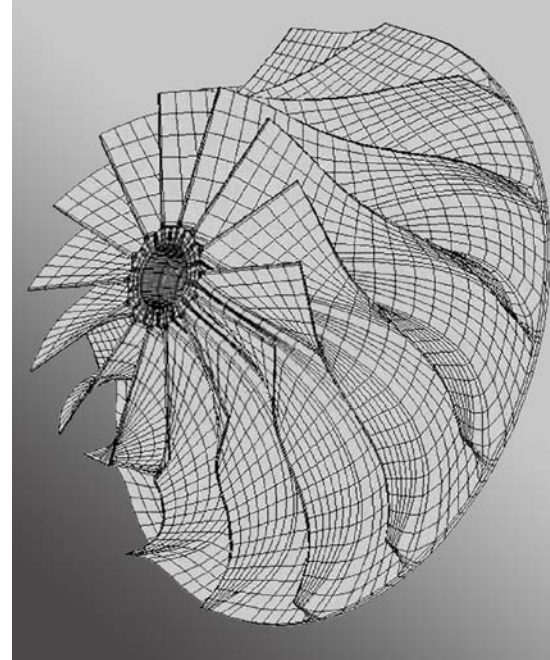
By utilizing the CAD input, ease of use, and meshing capabilities of ANSYS Workbench BladePro has the capability to work with CAD models.

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.

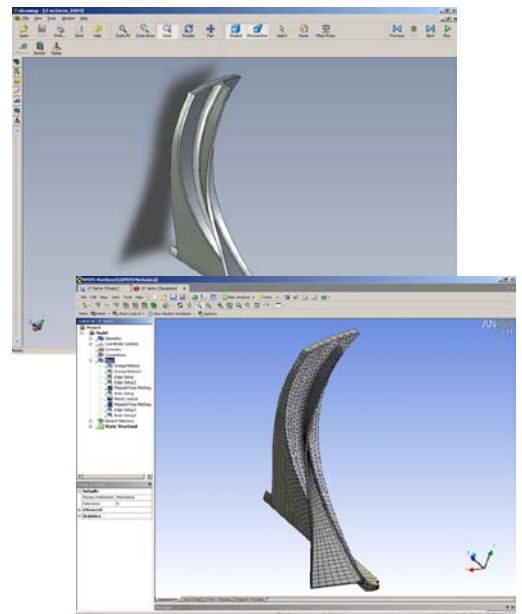
## Boundary Conditions - User Controlled Boundary Conditions

**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](http://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

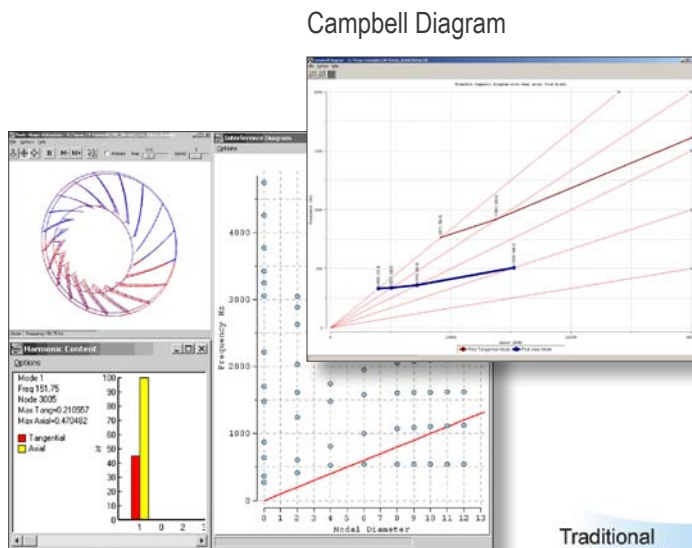
**Georgia Office**  
75 Fifth Street NW, Suite 312  
Atlanta, GA 30308  
Phone: 404.526.6188  
Fax: 404.526.6189

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

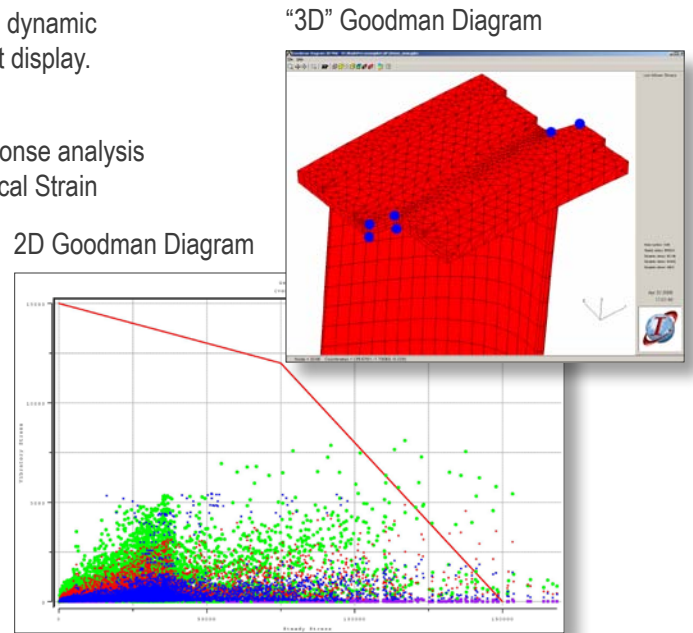
## Dedicated Post-Processing

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.

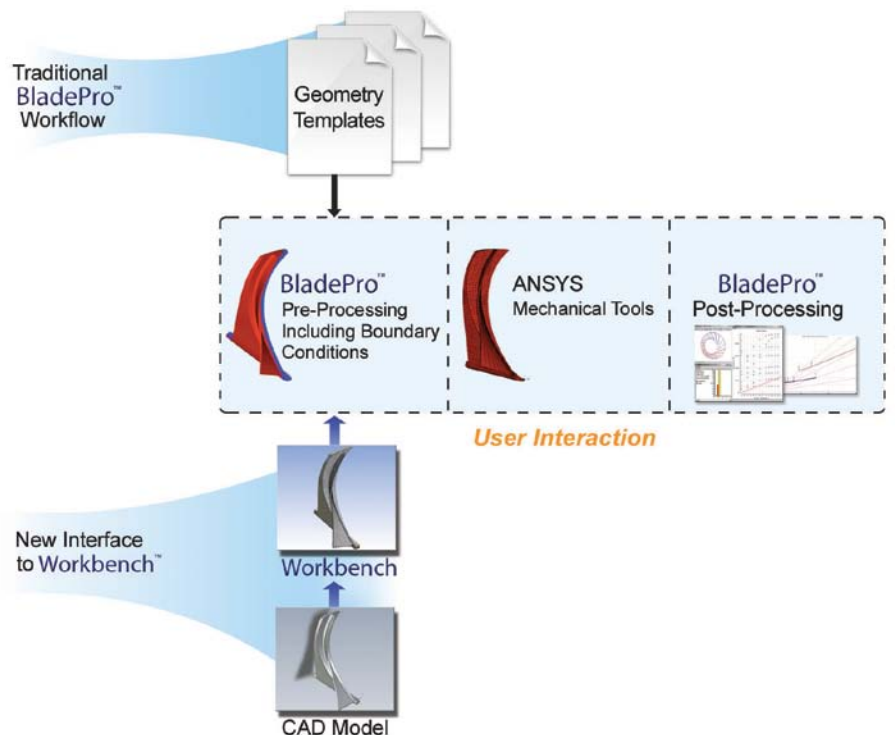


Animated Mode Shapes, Harmonic Content Display, Interference Diagram



2D Goodman Diagram

"3D" Goodman Diagram



ANSYS® is a registered trademark of ANSYS, Inc.