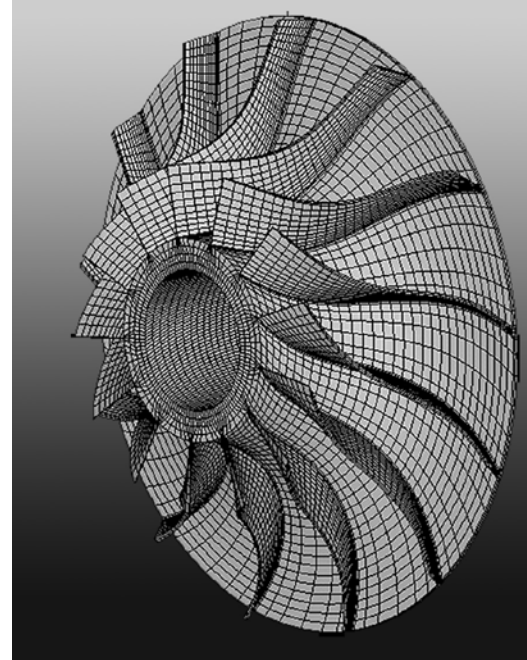


Significantly reduce the time required for performing structural analysis of turbomachinery components.

By utilizing the FEA power of ANSYS®, BladePro-CF™ significantly reduces the time required for performing structural analysis of radial flow impeller/compressor impellers. By acting as a companion product to ANSYS, the program assists the user in all aspects of radial flow wheel analysis: model generation, boundary condition application, analysis options, job submission, post-processing, and life assessment. Designed and written by engineers with years of experience in performing radial flow wheel analysis using ANSYS, BladePro-CF's menu system directly interacts with the ANSYS interface in a fashion that allows it to be used by those with no FEA experience, and by ANSYS experts alike. Even for experienced ANSYS users, the time it takes to produce results can be reduced by more than 50%.

Parametric Geometry Templates

By providing geometry templates for different components of a wheel model (blade, shroud, splitter, hub), BladePro-CF is able to exploit the capabilities of ANSYS and allow the user to focus on analysis results rather than modeling details. Through the GUI, the user has direct control over the mesh.

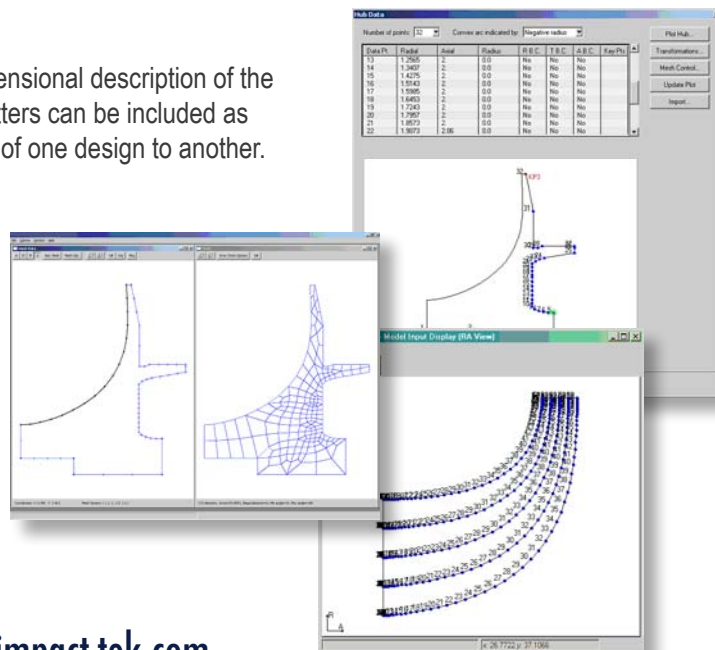


Blade Input

BladePro-CF allows users to input or import the three dimensional description of the blades along streamlines from the hub to the outside. Splitters can be included as well. A full set of algebraic operations allow for the scaling of one design to another.

Hub Geometry

The user may input or import hub geometry and has complete control over the mesh in the hub if he or she chooses. The user may also use the default auto-meshing option. Shroud Configurations Shrouds can be described to be included in the analysis. A three dimensional representation of the shroud where the blade meets the shroud can either be input or imported by the user.



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

Model Generation

FEA models are automatically generated, by BladePro-CF, which are optimized for both static and dynamic analyses and fed directly into ANSYS. BladePro-CF takes care of all the file handling, automatically. The model can be of a single sector using cyclic symmetry or of a full wheel. The model is suitable for steady stress analysis, natural frequencies and mode shapes, and forced response. Boundary conditions are applied to the model based on user input.

Analysis Options

The program supports steady stress analysis, natural frequency analysis, harmonic response, and dynamic stress calculations. For the expert user, the full set of ANSYS solution capabilities is available. A user-configurable material database provides for temperature dependent mechanical and fatigue properties. Fatigue calculations, to predict time to crack initiation, are performed either with the local strain approach or the Goodman diagram.

Thermal Effects

Using a series of ANSYS macros BladePro-CF allows users to specify initial shrink fit of their impeller on a shaft and calculate a steady-state temperature distribution to see the separate and combined effects on steady stress.

Dedicated Post-Processing

BladePro-CF provides automated tools for taking the results of a modal analysis and creating Campbell and Interference Diagrams. Also, the full suite of ANSYS post-processing options are available for examining stresses and mode shapes.

Combining these results with those from a static analysis and harmonic response analysis directly leads to the creation of a Goodman diagram.

BladePro-CF supports Windows PC's for either 32-bit or 64-bit machines.

