BladePro™: An ANSYS-Based Turbine Blade Analysis System
Avinash V. Sarlashkar
Girish A. Modgil
Mark L. Redding
Impact Technologies, LLC, Rochester, NY 14623, U.S.A

Abstract
This paper summarizes the architecture and capabilities of BladePro, an ANSYS based turbine blade analysis system with extensive automation for solid model and F.E. model generation, boundary condition application, file handling and job submission tasks for a variety of complex analyses; the program also includes turbomachinery specific post-processing and life assessment modules. BladePro is a cutting-edge example for vertical applications built on the core ANSYS engine using ANSYS APDL and Tcl/Tk. Examples of how the program makes effective use of the ANSYS preprocessor to mesh complex turbine blade geometries and apply boundary conditions are presented using specific examples. A real world application is used to demonstrate the pre-processing capabilities, static and dynamic stress analyses results, generation of Campbell and Interference diagrams and life assessment. The principal advantage of BladePro is its ability to generate accurate results in a short amount of time, thus reducing the design cycle time.

Introduction
Over the years, ANSYS has evolved into a very powerful FE-based analysis system to suit a variety of engineering applications in specific areas such as structural and thermal engineering, electro-magnetics, acoustics, computational fluid dynamic analyses, etc. ANSYS has rapidly become one of the most widely used simulation and analysis software. Selection of ANSYS as the core engine for development of vertical application such as BladePro is driven by many of its strengths. Some of the important strengths are: a) robustness and ability in generating and meshing complex geometries driven by parametric inputs, b) "low level" information for the solid and FE model entities available to the end user through APDL calls, c) support for a variety of physics environments, d) continuous development of fast and efficient solvers capable of handling large problems, and e) minimal effort involved in porting an ANSYS vertical application to a variety of hardware / operating system combinations.

Ever since the evolution of FEA, there has been a continuous and growing need for a powerful design analysis tool in the power generation industry. In general, turbines represent a class of challenging mechanical prime movers where steady and transient stresses (mechanical and thermal), turbine blade vibrations, and the start/stop cycling of the machines present interesting design challenges to produce a highly reliable machine with long design lives. These design lives may be as long as 20-25 years (150,000 to 200,000 hours of operation) for steam turbines, and as short as 3 years (25,000 hours) for certain components in a gas turbine. Of particular interest is the analysis of turbine blades, as these rotating components, if separated from their attachment to the rotor, have the potential for causing a tremendous amount of consequential damage, both in the form of human life and property damage. BladePro attempts to satisfy this need by using the industry-leading FE analysis platform to provide a menu driven, easy-to-use analysis tool. BladePro has been developed with two very broad areas of interest in mind.

1. Basic Design Analysis: Provide the turbine blade designer with a user-friendly interactive interface that allows for rapid model generation and access to a variety of analyses options in a typical design process for turbine blades. Designers can obtain the magnitude and distribution of static stresses the blade would be subjected to under operating conditions and possibly modify the design at the very nascent stages of the design cycle thereby reducing costs and time delays associated with changes late in the design cycle and in some cases after field testing.
2. **Advanced Vibration and Lifing Analysis**: These advanced analysis options provide the turbine blade designer with turbomachinery-specific analysis tools required for modal and forced response analyses, contact analysis, post-processing tools such as Campbell and Interference diagrams and Goodman diagrams. All these analyses areas represent essential ingredients in the turbine blade design process to either produce reliable designs or to better understand the cause of failures or to predict the remaining life of an existing design.

As the developer of BladePro, Impact Technologies has over 50 years of combined experience in analytical and practical life evaluation, design audits and failure analysis of turbomachinery.

**Leveraging the ANSYS Architecture**

The ANSYS architecture is very versatile and quite open in terms of access to low-level information in the ANSYS database. The following specific features of the ANSYS architecture allow for the development of a vertically integrated application such as BladePro that can be tightly integrated with ANSYS and at the same time can be run as a stand-alone application with limited capabilities.

1) **Command-line interface to ANSYS.** This is the most fundamental ANSYS facility that is key to "driving" ANSYS either through input files, macros or other interfaces such as Tcl/Tk or UIDL.

2) **Use of parameters in the form of scalars, arrays, and character strings as arguments to all ANSYS commands.** Again, this is another fundamental ANSYS feature allowing the generation of "parametric" models. Although at the present time, the ANSYS solid models are not parametric in a true sense, i.e., the input command sequence needs to be run in its entirety if changes are to be made to even a single dimension in a given model, it offers a very viable approach in generating models very quickly, even on inexpensive PCs. For example, once the BladePro input file is in place, a typical BladePro model generation phase that includes generation of solid model, FE model, and boundary conditions takes typically between 1-3 minutes on a 1000 MHz PC running Windows NT or Windows 2000.

3) **Macro facility within ANSYS.** Macros are an essential part of producing a vertical application that is highly modularized. For example, BladePro currently incorporates over 300 ANSYS macros. Equally important is the facility within ANSYS to encrypt the macros for distribution within a commercial package such as BladePro.

4) **Built-in Tcl/Tk interpreter.** This is an extremely important facility that allows rapid development of the graphical user interface (GUI) using the industry-standard Tcl/Tk programming language. It is the use of third-party language that makes possible the stand-alone use of BladePro features that do not depend upon ANSYS enabling the freeing up of an ANSYS license for non-ANSYS features. The versatility of Tcl/Tk combined with the power of APDL, makes it possible to create tools that are capable of automatic generation and analysis of highly complex geometries (such as turbine blades) with minimum user inputs.

**BladePro: General outline**

Figure 1 shows a snapshot of the ANSYS GUI with the BladePro button inside the ANSYS Toolbar, and the BladePro main toolbar that is launched after clicking on the BladePro button. The user interacts with the BladePro main menu to a) Enter blade geometry, material properties and associated necessary input defining a turbine stage, b) Define and initiate various analysis options, c) Perform a variety of post-processing operations.
Figure 2 shows a sector model for a typical blade design with an integral shroud. Several different components are labeled. Design-specific features of this blade design are an integral cover (shroud), and a slanted axial entry firtree style root (dovetail attachment).

This model was generated using BladePro by entering data directly from an engineering drawing through the BladePro GUI. As shown in figure 1, the main menu contains five menu items described below:

**File Menu:** Used for saving and reading in bpr files (the native BladePro input file). Another feature allows the user to merge data of different components from different bpr files into a single file.

**Model Input:** All the geometry associated with the different components is entered under this menu that contains submenus for cover (shroud), tenon, airfoil, platform, tiewire (mid-span shroud), root and disk data. Stage information such as blade count, speed of rotation, blade grouping, material properties and
aerodynamic forcing is also entered here. This menu also contains a comprehensive material database that is used to assign material properties to the different components.

**Generate Model**: Through this menu, the user can generate 2D and 3D models, calculate section properties of individual components and display boundary conditions (BC's). During model generation, BladePro will assign BC's such as couplings, displacement constraints, aerodynamic forcing and master DOF's by either using default (internal) settings or through user input. The information is written out to a free-format file that can be easily edited if the BC's need to be customized for a particular application.

**Analysis**: The FE model generated can be submitted for an analysis. BladePro supports the following types: steady stress, modal analysis, harmonic response, dynamic stress, thermal (steady-state and transient) and a 2D attachment analysis. Analysis options, amongst other things, allow the user to control the type of solver, number of processors, large deformation effects and frequencies of interest for modal analyses.

**Post-Process**: Tools include stress and displacement postprocessing, modeshape animation, Interference diagram, Campbell diagram, Goodman diagram, local-strain fatigue, and 2D attachment analysis postprocessing. These are described later in this paper.

**Help**: Brings up a detailed browser-driven context-sensitive help with keyword search capabilities.

In addition to the functionality offered by the BladePro main menu, the user always has the option to use the ANSYS command line to perform any of the standard ANSYS operations.

**Model Input**

The input format for a few of the significant components with their related features will be discussed here.

**Airfoil**: Figure 3 shows the airfoil input dialog box. Axial and Tangential points input by the user define the airfoil section geometry. Up to 100 points can be defined for each airfoil section with no restriction on the number of airfoil sections. Airfoil sections can be viewed individually and displayed with section property calculations or multiple sections can be stacked and viewed together. The airfoil template also gives the user the option of interactively adding or deleting airfoil input points. Sections can be scaled, rotated, and translated to allow for rapid analysis screening of design modifications.

Airfoil geometry profile can be input either as a series of points in which case a spline is fitted through the points (Figure 3) or by a series of arcs.
**Root:** Multiple templates exist for various blade roots. Figure 4 illustrates the dialog boxes and graphical display for the axial entry type. Other root types supported are straddle mount, finger root and T-root. Like the airfoil, the graphical display is interactive and gets updated with changes in root attachment geometry. Element size and number control is provided for mesh refinement or the internal default can be used.

![Figure 4 - Dovetail Input](image)

**Disk:** Geometry can be defined for symmetric and asymmetric disk profiles. BC's and element divisions can be specified for each input point, or default settings can be used. Figure 5 shows a disk profile defined where the points at the disk bore are constrained in all three directions.

![Figure 5 - Disk Input](image)
Cover/Shroud: Integral shrouds and those attached with tenons can be modeled. Figure 6 (left) illustrates an integral shroud being defined in two different planes (axial-tangential and radial-axial planes). The figure on the right shows a sample mesh for a multiple tenon configuration. The detailed fillet modeling allows for accurate stress prediction in this critical region. Blade-to-blade connectivity information is handled automatically and various blade group configurations can be analyzed. The angle on the airfoil tip (if any) is defined by the radial coordinates of the upstream and downstream points entered in the cover data template.

**Figure 6 - Shroud Configurations**

Wheel Configuration:

The Wheel Configuration Data dialog box (Figure 7) allows definition of blade-to-blade connections at cover and one or more tie-wires. Connections can be defined only when at least a cover (blue) or a tie-wire (red) is present. Edit fields allow the user to select the start and end blade number. User can select the number of blade groups to define and also the number of blades that are in each group.

**Figure 7 - Wheel Definition**
Material Database: BladePro contains a comprehensive material database with mechanical and thermal, fatigue and Goodman material property data sets (Figure 8). The database can be customized to contain user-defined materials and can be populated from third-party sources. Material properties can be plotted as a function of temperature and each component can be associated with a reference temperature at which material properties will be calculated during model generation and analysis.

Figure 8 - Material Database

Miscellaneous modeling features:
1. The user has direct control over the mesh refinement at both global and component levels.
2. Scaling, rotation and transformations can be applied at both global and component levels.
3. Both English and SI units can be used. Toggling from one to the other automatically reassigns material properties to the correct system.

By providing geometry templates for different components of a blade model, BladePro is able to exploit the solid modeling capabilities of ANSYS. The templates that have been developed using Tcl/Tk take the information from the GUI and write it to the bpr file that is subsequently read in by the macro library to parametrically generate blade components. The macro library makes extensive use of ANSYS boolean capabilities to create highly complex blade geometries with the closest degree of precision.

Design Aspects in Turbine Blade Design

In this paper, results of analysis for a 15” blade design are presented. This blade represents a typical design for a last stage on a medium-sized gas turbine or a more modern steam turbine. The design uses an axial-entry blade dovetail with three hook pairs. The shroud portion is integral with the blade that is preloaded at assembly.

Through the BladePro GUI, the designer can simulate a variety of operating conditions by specifying different boundary conditions and wheel configurations. BladePro makes use of the underlying ANSYS engine to:

- Perform Static Stress analysis with the option to incorporate spin softening and stress stiffening effects.
- Perform Modal analysis of a cyclic sector or full wheel model.
• Assign temperature dependant material properties.
• Apply aerodynamic and centrifugal loads.
• Simulate large deformation effects for long blades.
• Perform harmonic response analysis using modal super-position.

A design analysis is usually performed to identify potential resonance conditions during startup and operation and also to check whether static and dynamic stress distributions fall within design limits. If possible, a modal test could be performed on the blade row to determine its standing blade frequencies which can then be compared with frequencies calculated in a zero rpm modal analysis in BladePro (Figure 9). The comparison illustrated in Figure 9 is for a blade geometry similar to the 15" blade design used in this paper as an example. The graph is a plot of normalized frequencies vs. nodal diameters (harmonic numbers). The minimum and maximum percentage differences between theoretical and test results were found to be 0.2% and 4.8% respectively.

Once a good correlation is established (up to 5% difference in predicted vs. measured frequencies), the BladePro model can be submitted for at-speed frequency and forced (harmonic) response analyses. As described later in this paper, the results of at-speed frequency and forced response analyses are reviewed using the Campbell and Goodman diagrams.

![Correlation of 0 RPM modal test with BladePro results](image)

**BladePro Analysis Options**

The analysis options offered by BladePro include:

1) **Static Analysis.** The static analysis of a single-blade sector model or of a blade-group sector model can be performed. The former analysis option would be applicable to designs involving either freestanding or integrally shrouded designs common to gas turbine designs. The latter option is applicable to traditional steam turbine design practice. Loading options include centrifugal and aerodynamic forcing.
2) **Modal Analysis.** Standing and at-speed frequency calculations (modal analysis) for cyclic, as well as, non-cyclic wheel configurations are available. Modal analysis for the cyclic wheel configurations use the well-established solution techniques where several frequencies and mode shapes are calculated for a given set of nodal diameters (inter-blade phase angles). BladePro also supports modal analysis for wheel configurations that are not cyclic. This situation is more common to the conventional steam turbine design practice where blade groups can be of different lengths and/or certain features such as shrouds and tie-wire may be staggered or overlapping. The three Eigen solution methods for symmetric matrices available within ANSYS (Reduced, Block Lanczos, and Subspace) are all supported by BladePro. The experience gained by Impact Technologies personnel over many years in analyzing a large variety of turbine blade designs has played a major role in implementing the best modeling techniques for conducting modal analysis of turbine blades.

3) **Harmonic Response.** Harmonic response analyses for cyclic and non-cyclic wheel configurations are also supported within BladePro. The harmonic response calculation uses the mode superposition method, and therefore, a modal analysis prior to a harmonic response analysis is required. The harmonic response analysis allows the user to "freeze" the response of the turbine blade at a specific phase angle within a single cycle of vibration. This allows for a complete visualization of vibratory or dynamic stress distribution throughout the blade during a complete cycle of vibration.

4) **Fatigue Analysis.** The fatigue analysis tool calculates fatigue life using the strain-life approach which has been found to be appropriate for situations involving significant plastic deformation due to stress concentration. The user can define low and high cycle fatigue loads. The tool uses techniques such as rain-flow counting method for identifying damaging events and one of two methods (Morrow's or Manson & Halford) to incorporate mean stress effects on fatigue life. A linear damage summation technique is used to sum the damage due to different loading cycles.

All the necessary boundary conditions for the variety of analysis options discussed above are generated by BladePro during the model generation phase. Therefore, the user has to provide minimal information to initiate any of the analyses discussed above. For example, as a minimum, initiating a static analysis of a blade sector would involve specifying the shaft speed. Of course, there are additional controls that user can set. Some of the analysis options discussed above provide a "restart" capability wherever applicable to reduce solution times. It should be pointed out that for all analyses options, the user interface allows specification of ANSYS solution controls such as the selection of the ANSYS solver, the number of processors to use on multi-processor shared memory architecture machines, etc.

**BladePro Performance and Solution Times**

As far as run times are concerned, a static analysis for a typical BladePro model takes up to 5 minutes on a 1000 MHz PC running Windows NT/2000. We prefer the use of the Sparse Solver within ANSYS which experience has shown to be quite robust. The BladePro GUI does provide access to the frontal solver and the PCG solver. The frontal solver is perhaps the most robust solver within the ANSYS family of solvers, but comes with a penalty of high disk storage requirement for models with wavefronts in excess of 2500. BladePro provides access to Eigen value solutions using all three methods: The Reduced method, the Block Lanczos method (LANB) and the Subspace method. We have typically used the Reduced method and the Block Lanczos method. The Reduced method works well for most models where lower modes of vibration involving low disk participation are of interest. However, more complex modes, especially, with substantial disk participation are better calculated with the Block Lanczos method. The reduced method certainly offers the benefit of lower memory requirement (less than 200 MB for typical BladePro models) than the Block Lanczos method. The memory requirements for the Block Lanczos method can be significantly greater (upto 400 MB for typical BladePro models) when using the cyclic symmetry method. Finally, typical modal analysis runs where up to 20 nodal diameters may be calculated for up to 5 mode families can take up to an hour to calculate on a 1000 MHz PC with two processors running Windows NT/2000. We have found the overall performance of the Intel PC/Windows NT combination to be very acceptable for routine BladePro analyses.
Turbomachinery-Specific Postprocessing Tools

A typical mechanical design review of a turbine blade design for a fixed-speed machine involves design adequacy checks from a stress viewpoint (static and vibratory stresses) as well as from the viewpoint of placement of operating natural frequencies with respect to the multiples of machine running speed. For variable speed machines, vibratory stresses need to be calculated for all possible resonant conditions over the operating speed range of the machine. The generated vibratory stress information can be analyzed using a Goodman diagram approach as discussed later.

Campbell Diagram

Change in blade natural frequencies with respect to machine speed are plotted on a Campbell diagram (named after Wilfred Campbell, 1884-1924). Figure 10 shows the Campbell diagram for the 15" turbine blade used as an example. The Campbell diagram in this figure shows intersections of different nodal-diameter modes within each mode family (Axial, Tangential and Twist), with the respective per-rev lines. For blade designs where frequency tuning is a requirement, the points of mode family/per-rev line intersection will have sufficient margin with respect to the operating speed range.

Figure 10 - Campbell diagram for the 15" integrally shrouded blade design.
**Interference Diagram**

The Interference diagram is just another way to present the frequency and mode shape information. In the case of an Interference diagram, the nodal diameter vs. frequency information for different mode families is plotted at a given speed (whereas the Campbell diagram plots frequency as a function of speed). Figure 11 shows the Interference diagram for the 15" turbine blade used as an example. Shown on this figure are frequencies for different nodal diameter modes in the first three mode families. The proximity of the impulse line (shown in red) to any of the solid points shown in blue indicates the potential for resonance with a certain mode. If the impulse line passes close to any of the solid dots, (e.g., the solid dot with a square box for the first mode family), then the conditions for natural frequency and modal force coupling are simultaneously satisfied for a resonant condition.

*Figure 11 - Interference diagram @3600 RPM for the 15" integrally shrouded blade design*
Goodman Diagram

Mechanical adequacy of a design from the standpoint of static and vibratory stresses is typically evaluated using the Goodman diagram. The Goodman diagram shown in Fig. 12 has the mean or static stresses at critical locations plotted along the X-axis and the vibratory or the dynamic stresses at the corresponding locations along the Y-axis. Typically, the extreme limits for a failure envelope on a Goodman diagram are the material yield or ultimate strength along the X-axis, and fatigue strength in reverse bending (corresponding to specified number of cycles) along the Y-axis. BladePro can plot the mean and vibratory stresses for every single node in the FE model. The user has the ability to choose one or more vibration modes for plotting the vibratory stress component along with the mean stress component. For each node in the example model, the Goodman diagram in Fig. 12 shows the combination of static stress and vibratory stress for different modes of vibration.

![Goodman Diagram](image)

Figure 12 - Goodman diagram based on resonant stress for the first three modes of vibration

Analysis Results and Conclusions

Steady Stress Results: Figure 13 shows the distribution of static stresses throughout the blade. The static stresses are mostly the result of centrifugal load on the blade. A neck stress of 25,000 psi (172 MPa) is calculated for the blade dovetail, which is quite acceptable for a typical blade material such as AISI 403. Figure 14 shows a contour for displacement in the tangential direction. This contour plot provides information on the untwist in the blade at the mid-span location.
Campbell Diagram: Figure 10 shows the Campbell diagram for this 15" design. The Campbell diagram in this figure shows intersections of all the nodal diameters in the first three mode families. The nominal operating speed of 3600 RPM is shown on the right. This Campbell diagram shows that a resonant
condition is not predicted for the first two mode families. However, a mode in the third family may be resonant with a 15x excitation. The harmonic response analysis for this mode shows that the resonant stress for this mode is low (< 1000 psi or 6.89 MPa), and therefore, this resonance condition will be acceptable.

Interference Diagram: Figure 11 shows the Interference diagram for this 15” design. The Interference diagram is plotted corresponding to a nominal operating speed of 3600 RPM. The critical modes based on the proximity of the impulse line to different nodal-diameter modes in the first three families are identified as: 6-ND axial mode (Figure 15), 11-ND tangential mode (Figure 16), and the 15-ND twist mode (Figure 17). The harmonic response analysis for these modes was carried out using BladePro. The results for vibratory stresses in each of these modes are discussed in the next paragraph.

![Figure 15 - 6-nodal diameter axial modeshape](image)
Figure 16 - 11-nodal diameter tangential modeshape
Harmonic Response Analysis: Figures 18 through 20 show vibratory stress contours for the first three modes of vibration. The critical modes, 6 nodal-diameter axial mode (Figure 18), 11 nodal-diameter tangential mode (Figure 19), and the 15 nodal-diameter twist mode (Figure 20) were selected based on the Interference diagram (Figure 11). For the three modes, the amplitude of vibratory stresses is quite low, i.e., less than 3000 psi (20 MPa). This level of vibratory stress would be considered acceptable from a reliability standpoint for common turbine blade materials.
Figure 18 - Instantaneous resonant stress distribution in the 6-ND axial mode (psi)

Figure 19 - Instantaneous resonant stress distribution in the 11-ND tangential mode (psi)
Goodman Diagram: Figure 12 shows the Goodman diagram created with BladePro using the results obtained from the static and the vibratory analysis. It is quite clear from this Goodman plot that most of the nodes in the model are well within the Goodman envelope, and therefore, the design is considered acceptable. There are a few nodes that have large static stresses. In most cases, these would represent locations with small fillets, which are areas of high stress concentration. Generally, these locations will experience local plasticity, and therefore, will be typically investigated for low-cycle fatigue.

Future BladePro Enhancements

The BladePro product is constantly being updated to provide support for additional geometric features and customer-requested enhancements. One such ongoing effort is the interfacing with solid models generated by other third-party software. The ANSYS Connection product has proven to be an effective method of transferring solid models from various CAD systems into the ANSYS environment.

Additional analysis features that exploit ANSYS capabilities (e.g., contact analysis) are planned for future releases.

We expect that ANSYS enhancements to the solver technology, both in terms of speed and memory requirements, will benefit a BladePro user a great deal. In particular, we see the improvements to the Sparse Matrix solver in ANSYS 6.0 to benefit BladePro the most in terms of solution times and resource requirements. We expect the future releases of BladePro to benefit from the new features in ANSYS such as the Distributed Domain Solver (DDS) and the Probabilistic Design System (PDS in terms of the ability to use a probabilistic approach to designing turbine blades on a more routine basis.)
Summary

The broad base offered by the ANSYS architecture combined with the provision of using APDL macro and creating user-specific GUIs using Tcl/Tk, makes it possible to develop highly specialized, vertically-integrated ANSYS "plug-in" tools. By relying on programming languages (Tcl/Tk, C++, and APDL) that are supported on all platforms that ANSYS supports, BladePro can be made available to run within any ANSYS-supported hardware/operating system combination. Development of BladePro by Impact Technologies has been facilitated by years of experience using ANSYS for analysis and developing software for other power generation related applications. Additionally, BladePro is backed by over 50 years of combined experience in the field of turbomachinery design, analysis and testing. The program has been validated by comparison of analysis results to test data for more than a dozen different blade designs. The good correlation achieved is a testament to the accuracy of the ANSYS solvers and validity of the modeling techniques adopted in BladePro.

Acknowledgements:

Impact Technologies would like to thank Ben Atkinson and Ryuichi Machida, who are an integral part of the BladePro design and development team and without whose continued efforts all of this would not have been possible.

References

5. ANSYS 2000, ANSYS Online Documentation, ANSYS Inc.
7. Dello, J., Frequency Evaluation of a Steam Turbine Bladed Disk, Dresser Rand