TURBAERO SOFTWARE USER'S GUIDE

By Ronald H. Aungier
1211 Shady Hill Road
Greensburg, PA 15601
Phone: (724) 838-1223
Email: raungier@comcast.net

TURBINE AERODYNAMICS
Axial-Flow and Radial-Inflow Turbine
Design and Analysis
By Ronald H. Aungier

The *TurbAero* software implements the axial-flow and radial-inflow turbine aerodynamic design and analysis systems described in a book by R. H. Aungier (ref. 1). Some programs are also common to the compressor aerodynamic design and analysis system (*CompAero*) and so are also described in two other books by R. H. Aungier (refs. 2 and 3). This description will reference specific chapters from these references to relate the software to the descriptions they provides. The various design and analysis programs were written in Microsoft Visual Basic 5.0 for any IBM compatible PC using Microsoft Windows 95 or higher. They have been installed and used successfully under the Microsoft Windows 95, 98 and XP operating systems.   This description is specific to version 1.50, released for sale on 6/20/09.

Contents

A Typical Desktop Menu

1.0 Software Installation and Execution

Install the software by running program SETUP.EXE on the installation CD ROM and follow the on-screen instructions. The program setup process will create a directory TURBAERO on your hard drive (unless you choose another name) containing all executable programs, help files and support files required and a sub-directory called SAMPLES (e.g., C:\TURBAERO\SAMPLES) containing a set of sample input files for the various programs.

You can run all of the software by executing program TURBAERO.EXE, from which you can run the other programs. You can choose "Run" from the main menu and select the program that you want to execute or click the icon of the program you want to execute. The program's help menu will assist you in setting up a compact desktop menu (such as the one illustrated above). You can also run the programs individually using any of the usual Windows approaches, such as from shortcuts or the Windows Programs menu. But launching program TURBAERO from a desktop shortcut is usually the most convenient way to run them, particularly when alternating between several programs during the design process.

On the first run of any application, you should select SetUp from the application's main menu to set your printer margins, font and font size, screen font, font size and colors, default units, etc. The on-screen help should be more than sufficient to carry out this setup process. You should note that some printer drivers impose margins external to the setup here to avoid printing in certain areas of the page. You may have to experiment a little to get the margins you want. The information generated during the setup process is stored in the Windows registry by the Windows operating system for use by all TurbAero applications.

2.0 Features Common to Most Programs

The various programs use a fairly standard Windows type File menu with the usual Open, Save, SaveAs and Close options. This menu will also contain some of the most recent input files you have used previously. Most programs also offer an input file input box, where you can type in the path and input file name directly. Each Program uses a recommended file name extender. You don't have to follow that practice in most cases, but file searches will be much easier if you do. Often when designing, one switches from one program to another, using the same base input file name for each with the default extender for each program. When creating input files for one program for use in another, the default assumption is that you will be doing this, but again you can over ride the default if you want to.

Except for programs BEZIER, GASPATH and RIGPAC, (which use dimensionless data), all programs have a general input-output units capability. You can select the units you want to use from a rather large number of choices. The *TurbAero* system always maintains a default units set, which is assumed to be the one you want when you start a new case. You can set the default units from any program using the units package from the SetUp menu in the main window. You can also customize the default units set from other programs, and you can choose different units for any particular problem. Input-output units are saved in all input files to be sure the correct units are used for each problem. You can change the units used for a problem after input data has been entered, but that will not change the input data. You will need to edit the input data if it needs to be converted to the new units. When you are setting the units, the first step is to select the standard set that is closest to what you want (English, SI-metric, etc.). Then you can customize individual units to obtain the combination that you want. Some units are derived from your selections (e.g., enthalpy from energy and mass). The derived units are displayed as you make your selections. Usually, a checkbox will be visible to permit you to use the selected units as the *TurbAero* system default set.

The various programs that offer aerodynamic analysis employ a general ideal/non-ideal gas equation of state package, including a gas property database currently containing over 100 compounds. The process of selecting a gas or gas mixture equation of state will be identical in these various programs. Several options are available:

- Load Pseudo-Perfect Gas Data: Here you load the gas mole weight, specific heat, Cp, isentropic exponent and viscosity constants for a linear fit with temperature. The gas is over-specified to permit you to approximate a real gas with this pseudo-perfect gas model as described in chapter 2 of refs. 1, 2 and 3 and in ref. 4.
- Ideal/Non-Ideal Gas Database: Uses a gas property database to set up gas or gas mixture equation of state. You can use an ideal gas [with Cp = Cp(T)] or one of the two available Redlich-Kwong non-ideal gas models.

When the second option is used, you can select an option to set up a pseudo-perfect gas model. The pseudo-perfect gas model offers very fast computations and the database makes it simple to create the model for a gas or a gas mixture. The program calculates the constants that you would otherwise enter as input data under the first option. Indeed, it is faster than the ideal-gas model under the second option, since it uses constant specific heats. It is adequate for nearly any internal flow analysis and has been used quite successfully for most compressor applications. But it is not recommended for turbine performance analysis applications, since it can compromise the prediction accuracy, particularly for operation near the vapor saturation line or with two-phase flow. See section 18.0 for a more complete discussion of important issues relative to fluid thermodynamic modeling and the equation of state. In certain turbine applications, these issues can become very significant. They can be very confusing to the user who does not understand them. This typically leads to a fatal error in an analysis with no indication of its cause.

A good starting point for new users is to run Program RKMOD, which performs a variety of thermodynamic equation-of-state calculations. Follow the menu and screen instructions carefully and you should have little trouble mastering the process of setting up a new gas thermodynamic model. Since it is needed so often and by so many of the programs, it is a good idea to understand the process early.

3.0 Available Aerodynamic Design and Analysis Programs

In addition toTURBAERO.EXE the following computer programs are available for centrifugal and axial-flow compressor aerodynamic design and analysis.

- AXSIZE.EXE: an axial-flow turbine stage preliminary aerodynamic design program (ref. 1, chapter 6).
- AXTURB.EXE: an aerodynamic performance analysis for single-stage or multi-stage axial-flow turbines (ref. 1, chapters 4 and 5).
- AIRFOIL.EXE: an axial-flow turbine airfoil detailed aerodynamic design program (ref. 1, chapter 7).
- RIFTSIZE.EXE: a radial-inflow turbine stage preliminary aerodynamic design program (ref. 1, chapter 10).
- RIFT.EXE: an aerodynamic performance analysis for single-stage or multi-stage radial-inflow turbines (ref. 1, chapter 9)
- RIFTNOZ.EXE: a detailed aerodynamic design program for radial-inflow turbine nozzle rows (ref. 1, chapter 11).
- GASPATH.EXE: a general gas path (end-wall contours and blades) design program (ref. 1, chapter 11 and ref. 2, chapter 7). Primarily used for radial-inflow turbine impeller design.
- FLOW3D.EXE: a quasi-three-dimensional inviscid flow field analysis (chapter 14 of ref. 1, chapter 12 of refs. 2 and 3) with supporting surface boundary layer analyses (ref. 1, chapter 13, ref. 2, chapter 15 and ref. 3, chapter 5).
- B2B2D.EXE: a two-dimensional blade-to-blade potential flow analysis (ref. 1, chapter 13, ref. 2, chapter 13 and ref. 3, chapter 5) with supporting surface boundary layer analyses (ref. 1, chapter 13, ref. 2, chapter 15 and ref. 3, chapter 5).
- TDB2B.EXE: a time-marching blade-to-blade flow analysis (ref.1, chapter 13, ref. 2, chapter 14 and ref. 3, chapter 5) with supporting surface boundary layer analyses (ref. 1, chapter 13, ref. 2, chapter 15 and ref. 3, chapter 5).
- RKMOD.EXE: an ideal/non-ideal fluid equation-of-state package for a range of thermodynamic property calculations (refs. 1, 2 and 3, chapter 2). This same equation of state package is used by many of the other programs.
- GASDATA.EXE: a gas property database maintenance program to support program RKMOD and the other programs that use its equation of state.
- RIGPAC.EXE: (Radial Impeller Geometry PACkage): a general geometry package primarily for radial-inflow turbine impellers, but suitable for other vaned components also. It performs many operations and geometry calculations commonly used in centrifugal compressor design and implementation. Conceptually, it is similar to Program GASPATH, except that it starts from known geometry (e.g., to treat existing component designs). The value of this type of program is discussed in chapter 11 of ref. 1.
- BEZIER.EXE: a program to generate the curves used by Program GASPATH to construct the end-wall contours and blades (ref. 1, chapter 11 and ref. 2, chapter 7). It can be accessed directly from Program GASPATH. But, on occasion it is convenient to work directly with this program for creating or refining curves just to avoid the need to navigate through the menu structure of Program GASPATH to get to this sub-program.
- EXHAUST.EXE: a turbomachinery exhaust diffuser aerodynamic performance analysis (ref. 1, chapter 12 and ref. 3, chapter 13).
- VOLUTE.EXE: a volute geometry detailed design program suitable for centrifugal compressors and radial-inflow turbine stages. It accomplishes detailed passage designs at specified circumferential stations using either elliptical or rectangular passage cross-sections for external and internal volutes.
- BLADE.EXE: a blade-section geometry program for the various axial-flow compressor standard airfoils used by *CompAero* (ref. 3). Its original purpose was to check out the blade geometry routines and database used by several programs. Some users have found it useful for generating blade section geometry for simple wind-turbine applications, so it has been included in *TurbAero* also.

Included in the distributed files is file RKDBASE.DAT, which contains gas thermodynamic data for over 100 compounds collected over several years. This is supplied as a courtesy to the user, who must use the data at his or her own risk. The fundamental technology involved prohibits any general statement or conclusion regarding either its accuracy or its adequacy for the user's range of thermodynamic conditions. Indeed, one function of Program RKMOD is to permit users to check representative calculations against other sources to evaluate the accuracy of the data and the equations of state for specific problems of interest. Program GASDATA can be used to maintain this database, print the property data stored, add new compounds, edit existing data, etc.

4.0 Program AFTSIZE

Program AFTSIZE implements the preliminary design procedure described in ref. 1, chapter 6. It uses basic design correlations and design criteria to rough out a "first cut" axial-flow turbine stage design. Its purpose is to convert your design objectives into a viable stage configuration to provide a starting point for detailed component design activity. Program AFTSIZE creates the input data files for other axial-flow turbine aerodynamics programs needed for the detailed stage design, including Programs AXTURB and AIRFOIL. Some editing of those input files is often necessary. Scaled blade thicknesses and recommended surface finish values may not be appropriate for manufacturing or mechanical design considerations. With this brief introduction and the fairly extensive on-screen help available from Program AFTSIZE, it is a fairly easy program to learn to use.

5.0 Program AXTURB

Program AXTURB provides a hub-to-shroud (multiple-streamline) aerodynamic performance analysis for single-stage or multi-stage axial-flow turbines using the procedures described in ref. 1, chapters 4 and 5. For new designs, the initial input file for this program will normally be supplied by Program AFTSIZE. Indeed, Program AXTURB directly supports Program AFTSIZE by supplying a basic evaluation of the performance potential of the preliminary stage design. As the design progresses through the detailed design process, the various detailed design programs have the capability to continually update the input file for this program with the refined geometry data. Users will have to enter all input data only for turbine stages not designed with the *TurbAero* system. The on-screen help describes the input data required, including a number of pictures for illustration.

This program can collect and save performance prediction results and use them to display or print performance maps for the turbine. Three maps are provided: a mass-flow type, an efficiency type and a power type. The map menu in the main window lets you select the variables to map, the curve legend (e.g., to identify alternate speed lines and plot them as separate curves) and let you transfer to the Map window to load or edit data, including test data points if available. When map options are set up, the main window shows a check box to let you save (when checked) or not save (when unchecked) subsequent performance predictions in a map data file. When predictions have been saved in the map file, they can be saved in the input file for use on future runs from the current input file. The on-screen help describes other features you need to know about this general map capability. It allows you to collect results from several performance analyses to map the complete turbine performance in a variety of common formats. You can also export the map data to an ASCII text file for subsequent import to spreadsheets or graphics software. For added support, the program also lets you export a variety of turbine-stage or overall turbine performance data to an ASCII text file for subsequent import to spreadsheet or graphics software. However, this general ASCII export data is not saved when exiting the program. You must export the data before exiting or it will be lost. It also has a special option to export a performance map in files suitable for importing into Sinda/Fluint, a generalized fluid and thermal analysis for complex systems by C&R Technologies (see section 17.0).

6.0 Program AIRFOIL

Program AIRFOIL is a detailed aerodynamic design program for constant-radius airfoil sections to be stacked to define the axial-flow turbine blades as described in ref. 1, chapter 7. It is a modification of a method proposed by Ye (ref. 5). It uses basic inlet and discharge specifications from the desired velocity triangles and a series of optional surface reference points to define smooth airfoil shapes for the various sections. It makes extensive use of interactive graphics procedures to enable the designer to develop airfoils consistent with the design objectives. Usually, the original input file for this program is created by Program AFTSIZE. Since control of the blade throat-width to pitch (gauging angle) is usually essential, alternate options are provided to semi-automate the design process to greatly simplify it as compared to the original reference. Program AIRFOIL can create or update input data files for Programs B2B2D or TDB2B (geometry data portion only) to evaluate the blade loading for this airfoil section (note that this is a two-dimensional cascade blade loading evaluation for this constant-radius section only). You will need to use care in selecting the mass flow and stream sheet thicknesses used in the actual analysis to be sure they are consistent with each other. If discharge static pressure will be specified (TDB2B, only) that is not an issue. In general, blade loading evaluations close to the vapor saturation line or close to choke are not very useful or informative. See section 18, below, for suggestions for dealing with those cases. Program AIRFOIL can also export geometry data to an ASCII text file for subsequent import to drafting, manufacturing, spreadsheets and graphics

software. Multiple blade sections can be designed and saved in a single input file, which can then be stacked to define the complete blade. Program AXTURB can import the blade geometry from a Program AIRFOIL input file for a performance analysis after the design of all blade sections has been completed.

7.0 Program RIFTSIZE

Program RIFTSIZE implements the preliminary design procedure described in ref. 1, chapter 10. It uses basic design correlations and design criteria to rough out a "first cut" radial-inflow turbine stage design. Its purpose is to convert your design objectives into a viable stage configuration to provide a starting point for detailed component design activity. Program RIFTSIZE creates the input data files for other radial-inflow turbine aerodynamics programs needed for the detailed stage design, including Programs RIFT, RIFTNOZ, GASPATH (nozzle or impeller) B2B2D and TDB2B (nozzle) and FLOW3D (impeller). Some editing of those input files is often necessary. Estimated clearances and surface finish values may not be appropriate for manufacturing or mechanical design considerations (some editing is possible when exporting some of these input files). With this brief introduction and the fairly extensive on-screen help available, it is a fairly easy program to learn to use.

8.0 Program RIFT

Program RIFT provides a mean-line (one-dimensional) aerodynamic performance analysis for single-stage or multistage radial-inflow turbines using the procedures described in ref. 1, chapter 9. For new designs, the initial input file for this program will normally be supplied by Program RIFTSIZE. Indeed, Program RIFT directly supports Program RIFTSIZE by supplying a basic evaluation of the performance potential of the preliminary stage design. As the design progresses through the detailed design process, the various detailed design programs have the capability to continually update the input file for this program with the refined geometry data. Users will have to enter all input data only for turbine stages not designed with the Turb*Aero* system. The on-screen help describes the input data required.

This program can collect and save performance prediction results and use them to display or print performance maps for the turbine. Two maps are provided: a mass-flow type and an efficiency type. The map menu in the main window lets you select the variables to map, the curve legend (e.g., to identify alternate speed lines and plot them as separate curves) and let you transfer to the Map window to load or edit data, including test data points if available. When map options are set up, the main window shows a check box to let you save (when checked) or not save (when unchecked) subsequent performance predictions in a map data file. When predictions have been saved in the map file, they can be saved in the input file for use on future runs from the current input file. The on-screen help describes other features you need to know about this general map capability. It allows you to collect results from several performance analyses to map the complete turbine performance in a variety of common formats. You can also export the map data to an ASCII text file for subsequent import to spreadsheets or graphics software. Program RIFT can also update input files for Programs B2B2D, TDB2B and FLOW3D to supply gas model and inlet flow data not available from the detailed design programs. It also has a special option to export a performance map in files suitable for importing into Sinda/Fluint, a generalized fluid and thermal analysis for complex systems by C&R Technologies (see section 17.0).

9.0 Program RIFTNOZ

Program RIFTNOZ is a detailed aerodynamic design program for radial-inflow turbine nozzle rows as described in ref. 1, chapter 11. It uses a standard airfoil family adjusted to match the inlet and discharge velocity triangles. This is the usual approach to nozzle detailed aerodynamic design, although Programs GASPATH and BEZIER can also be used for the rare cases where greater generality is needed. The initial input file is normally supplied by Program RIFTSIZE, but can be created from scratch if needed. Program RIFTNOZ can create or update input files for Programs RIFT, B2B2D and TDB2B. If gas model and inlet flow data are supplied, it can also perform a linearized blade-to-blade flow analysis on demand for an approximate blade loading evaluation. It can also export detailed geometry data to an ASCII text file for subsequent import into drafting, manufacturing, spreadsheet or graphics software.

10.0 Programs GASPATH and BEZIER

Program GASPATH (ref. 1, chapter 11 and ref. 2, chapter 7) designs hub and shroud contours and blades (if

present) for turbomachinery components. Length units are optional. Blade angles are always specified in degrees with respect to tangent. Program BEZIER generates the curves used to construct the blades and end-wall contours. It can be run directly from Program GASPATH, or it can also be executed separately. This can be convenient when curve generation or editing activity is your primary interest, since you do not have to navigate through Program GASPATH's menus to get to it. Two input files are used, the basic gas path geometry file and the curve file. The latter input file has a specific restriction. Its base file name <u>must</u> be the same as that of the gas path geometry file, and its file name extender <u>must</u> be .BEZ so that Program GASPATH can find it. When the curve file is generated through Program GASPATH using the BEZIER module, this naming convention is handled automatically. But, if Program BEZIER is used to create it, the user must keep this naming convention in mind.

The most common use of this program is for radial-inflow turbine impeller design, but it can be used for almost any component. Program RIFTSIZE is normally used to create its input files. It has provision for creating or updating geometry input files for Programs RIFT and FLOW3D and to create an input file for Program RIGPAC. It also has a special provision for exporting the shroud contour coordinates to a file. This data can then be imported into Programs FLOW3D or RIGPAC to be imposed as a new shroud on the base data in use by those programs. This lets you use Program GASPATH to design alternate shroud contours to be used in deriving a family of impellers of various flow capacities or pressure ratios from a base impeller design

Becoming familiar with Programs GASPATH and BEZIER is more difficult than the other programs because so many options are offered. But the serious designer will soon find that every option is really needed. Extensive on-screen help is provided in these two programs for that reason. Users who take the trouble to master these programs will find they have a very powerful design tool. After the initial confusion is over the programs are actually quite easy to use. The program can design 4 blade types:

- 2D axial-element blades whose mean camberline is composed of axial line elements. You design a single blade angle distribution as a function of radius. These blade angles lie in a radial surface. The distribution must include the entire radial domain where blades exist.
- 2D radial-element blades whose mean camber line is composed of radial line elements. You design a single blade angle distribution as a function of the axial coordinate and for a specified radius. These blade angles lie in the constant radius surface and must include the entire axial domain where blades exist.
- 3D straight-line element blades whose mean camber line is composed of straight-line elements connecting specified points along the hub and shroud camberlines. You design the hub and shroud surface blade angle distributions. These blade angles lie in the hub and shroud surfaces, defined as a function of dimensionless (0 to 1) meridional distance.
- General 3D or sculpted blades, which are defined as a modification to the third blade type.

Ref. 1, chapter 11, section 11.2 provides relationships between the blade angle and polar angle for the first three blade types. The component geometry is constructed from curves defining the hub and shroud contours, blade camberline angle and blade thickness distributions. The curves are designed in a separate program module (BEZIER). They are saved in a curve file for use in constructing the component geometry. Curve design is independent from the gas path design. You label each curve and use these labels to select curves you want to use to construct the component gas path. Curve construction options include:

- Bernstein-Bezier polynomial curves (most general option).
- Specify individual data point coordinates along the curve.
- Use the largest circular-arc that matches specified end points and slopes (a line segment is added at one end, if needed, to complete the curve).
- Use a three-point spline fit of end points and slopes and one interior point.
- Use a 3rd order polynomial that matches specified end points and slopes.
- Composite curves formed by joining two or more other existing curves.
- Fit a Bernstein-Bezier polynomial curve to any existing curve of a different type.

The options three through five allow straight-line segments ending at the end-points to be included. The last option is particularly useful for converting initial curves supplied by Program RIFTSIZE to a form more readily modified to refine the preliminary design. You will select specific curves from the curve file for the contours and distributions that define the component geometry. Curves are numbered sequentially as you create them. Selecting a curve

defines the curve number to be used. Subsequent editing of curves won't affect your selection. But <u>if you delete curves (hence, changing numbers), you must reselect curves</u>. The normal sequence of gas path design steps is:

- Specify the basic design data (case title, number of blades, etc.)
- Develop the curve file you want for the design.
- Select the curves you want from the curve file.
- Construct the component geometry and evaluate.
- Edit the curve file, design data and/or curve selection to refine the design.

Specific considerations with respect to the hub and shroud contour design are:

- For all contour curves, you specify a data point distribution along the curve where specific coordinates are calculated and saved in the curve file. The number of data points used on the hub and shroud contours must be the same. These points are the quasi-normal end points for the gas path. For 3D blades, the straight-line elements of the blade camber surface lie in the quasi-normal surfaces.
- When candidate hub and shroud curves both exist, use the curve edit option to edit two curves at once. Then the plots provided show both curves on the screen to better guide their refinement. Tables of coordinates, slopes, curvatures and passage area data are very useful in evaluating these curves while in module BEZIER.

  Specific considerations relative to blade design are:

- Blades must start and end on quasi-normals (not necessarily the first and last ones). At some point, you must specify the quasi-normal numbers for the blade leading and trailing edges. This is done by editing the basic design data. The program will remind you to do this when hub, shroud and blade angle distributions have all been selected. If you change the number of quasi-normals (curve data points), revise these.
- For 2D axial-element blades, there is a special option for a camberline constructed with up to two circular- arcs rather than using the curve file.
- Remember that design blade angles are different for the 3 blade types. For 2D axial-element, they lie in a radial plane. For 2D radial-element, they lie in a constant (specified) radius surface. For 3D blades, they lie in the hub and shroud meridional surfaces. Also, remember 3D blade angle distributions are a function of dimensionless meridional distance (0 to 1).
- Blade thickness distributions are specified as a function of dimensionless blade camberline length (0 - 1). For 3D blades, it is specified on both hub and shroud and varies linearly along quasi-normals. For 2D blades one distribution is used, which is applied to all stream surfaces. You can use the curve file to create the thickness distributions or use a special option for constant thickness blades with a linear taper near the leading edge. When you need a more general thickness distribution for 2D blades, you may find it useful to complete the design and export it to program RIGPAC for final adjustment, since that program of offers more flexibility in thickness distributions on 2D blades.

Program GASPATH can create or update geometry data for input files for Programs RIFT, FLOW3D and RIGPAC. It can also export geometry data to an ASCII text file for subsequent import to drafting, manufacturing, spreadsheet or graphics software.

11.0 Program FLOW3D

Program FLOW3D provides a quasi-three-dimensional internal flow analysis and supporting end-wall and blade surface boundary layer calculations (ref. 1, chapter 14 and refs. 2 and 3, chapter 12). Program FLOW3D is used for radial-inflow turbine impeller design (ref. 1) and centrifugal compressor impeller design (ref. 2). In that role, it is a primary evaluation tool for the gas path design generated by Program GASPATH. When used in impeller design, its geometry input file data will normally be generated (and updated) by Program GASPATH. Program RIFT has the capability to supply all other input data required for an impeller analysis with this program. So, you will basically have no need to create or edit input files for this program when designing impellers. It has been extended to include axial-flow compressor applications, but most of its blade geometry options are primarily suited for impellers. Users rarely need to create or edit input files for this program, since other programs usually supply them. This program can, in turn,

provide an input file for the more exact blade-to-blade flow analyses programs B2B2D and TDB2B on individual stream surfaces after the quasi-three-dimensional flow analysis is completed. That may be useful to confirm that the linearized blade-to-blade flow analysis used in program FLOW3D has achieved acceptable accuracy. The blade types it considers are

- Three-dimensional camberlines formed by straight-line elements from hub to shroud.
- Two-dimensional camberlines formed by axial-line elements from hub to shroud.
- Two-dimensional camberlines formed by radial-line elements from hub to shroud.
- General three-dimensional camberline blades.
- Standard axial-flow compressor airfoil sections stacked in the radial direction.

The first three types are common blade constructions used in radial-inflow turbines and centrifugal compressors. The fourth is quite general, but also quite difficult to define without a separate blade design system, but certainly possible. Programs GASPATH and RIGPAC provide that capability for radial-inflow turbines and centrifugal compressors. The fifth type is directly relevant to axial-flow compressors. Indeed, the fifth type is just a built-in preprocessor that generates input data for the fourth type, using standard blade sections of ref. 3, chapter 4.

Input data for the fourth type are entered on a series of stream surfaces spaced across the passage. Stream surfaces can be spaced at specified fractional distances or fractional areas between the hub and shroud, or they can be constant radius surfaces. The data supplied on these surfaces are the normalized stream-wise distance from the leading edge, polar angles and blade thicknesses. The on-screen help describes the data requirements, but providing the data without support from a blade design system is obviously difficult.

It is not very difficult to create an input file for the axial-flow compressor blade option. The blade is defined by supplying blade sections at a series of radii using specifications similar to the previous discussion for the blade-to-blade flow analysis programs. But most users will find it much easier to set up the basic problem in program AXCOM, run that analysis and then export a program FLOW3D input file. That will supply the geometry and all other input data required for program FLOW3D.

A keyboard input and edit capability is included, of course. You may need to use this for components other than impellers or for existing impellers not designed by the *TurbAero* design and analysis system. In many cases, you may find it more convenient to model existing components in Program RIGPAC, which accepts a wide range of geometry input options and basically performs all of the functions of Program GASPATH for existing geometry. Program RIGPAC can set up an input file for this program to give you a simpler entry into the Turb*Aero* system for existing designs.

When running Program FLOW3D, two or more streamwise stations (quasi-normals) should be located upstream and downstream of the blade passage. So, include vaneless upstream and downstream extensions in Programs GASPATH or RIGPAC before generating or updating an input file for this program.

Program FLOW3D can also perform two basic design functions, both intended to develop alternate shroud contours for the resident gas path design. These functions are often used to generate a family of impellers having different flow capacities or pressure rations from the same basic design. One option is to impose a change in the annulus gross area (i.e., neglecting blade metal blockage) before conducting the flow analysis. You specify the fractional area changes at the blade leading and trailing edges and at mid passage to do this. The other is to perform a flow cut, which simply locates the streamline in the flow field for which a specified fraction of the total mass flow lies between it and the hub contour. This streamline would then be the new shroud contour for the reduced flow capacity impeller.

When alternate shroud contours are generated in Program FLOW3D, they can be exported to a file for subsequent import into Program RIGPAC. Similarly, alternate shroud contours previously exported from Programs GASPATH or RIGPAC can be imported into Program FLOW3D to be imposed as an alternate shroud contour on the resident input file geometry. When this is done, the alternate contour is saved in the input file so the import is done only once. When the analysis is complete, Program FLOW3D can create input files for one of the more precise blade-to-blade flow programs (B2B2D or TDB2B) for a final and more rigorous check on the blade loading. This can be done for any of the stream surfaces included in the quasi-three-dimensional flow analysis.

For the rare cases where input data will be loaded directly, a brief description of the data required will be given so the user will no what to expect. You will specify the number of streamwise stations (quasi-normals), stream surfaces, quasi-normal numbers at the blade leading and trailing edges and the number of full-length blades. You can specify one set of splitter blades, in which case you will specify the meridional locations of the splitter blade leading edge for both the hub and shroud contour expressed as a fraction of the meridional length of the full-length blades. You will enter the

angle corresponding to the direction of rotation (or positive swirl, if not rotating). The program uses this to set its blade angle convention, so define it in the angle convention you will use in your geometry input data (e.g., for impellers with blade angles with respect to tangent, enter 180°). The blade type is specified (the same blade types as used by Program GASPATH can be used). If using 3D ruled surface blades, the hub and shroud leading edge polar angles must be supplied to properly orient the blade. You will enter the mass flow rate, rotation speed and inlet conditions (total temperature, total pressure and either swirl velocity or flow angle). The inlet conditions can be entered as a single (constant) condition or as profiles. The axial (origin arbitrary) and radial coordinates for all quasi-normals on both the hub and shroud contours is required.

The blade data required is always blade angle (using the blade angle convention discussed above) and blade thickness. For 3D ruled surface blades, they are entered for both the hub and shroud at all quasi-normals in the blade passage, and the blade angles are measured <u>in the stream surfaces defined by those contours</u>. For 2D blades, data are entered as a function of the relevant independent coordinate (axial coordinate for radial element blades or radial coordinate for axial-element blades) and the blade angles are measured <u>in the surface defined by that coordinate</u>. Equations (7-1) of ref. 1 provide these definitions in analytical form for these three blade types. For radial element blades, you will also have to specify the radius at which your data apply. A special option is available to handle standard axial-flow compressor airfoil sections stacked in the radial direction.

The standard input/output units and gas equation of state packages are employed, but the analysis will use <u>only the pseudo-perfect gas model</u>. You will still have the convenience offered by the gas property database, but in the interest of practical computation speed, the full non-ideal gas model is not available. To permit running the axisymmetric, three-dimensional hub and shroud boundary layer analysis you will need to give information regarding portions of the end-walls that are rotating, force defect terms, etc. which should be easily handled by the on-screen guidance given. This boundary layer analysis is described in ref. 1, chapter 15 but was subsequently revised in ref. 2, chapter 12

12.0 Programs B2B2D and TDB2B

These two blade-to-blade flow analysis programs can all be run using the same input file. There is one precaution required. Program B2B2D is a two-dimensional potential flow method (ref. 1, chapter 13, ref. 2, chapter 13 and ref. 3, chapter 5) suitable for subsonic and slightly transonic Mach number levels. Program TDB2B is a general time-marching method (ref. 1, chapter 13, ref. 2, chapter 14 and ref. 3, chapter 5), suitable for any Mach number level. Due to its significantly longer computer running times, it is normally only used when Program B2B2D simply can't handle the Mach number levels involved. Also because of the longer running times, Program TDB2B has a provision to create a restart file from which an analysis that has been terminated can be restarted from the point where it left off. These issues are no longer very significant for modern PCs, which are generally fast enough to accomplish the analysis with quite reasonable computer running times.

The most common use of the two-dimensional programs is for nozzle blade loading evaluation (for program RIFTNOZ) and for a final blade loading evaluation for impellers to be sure the linearized method used in Program FLOW3D achieved sufficient accuracy. It is also useful for two-dimensional cascade blade loading evaluations for constant-radius airfoil sections designed by program AIRFOIL. In these roles, the input files are usually created (totally or at least in part) by those programs that they support. Both programs offer a blade surface boundary layer analysis (ref. 1. chapter 13; ref. 2, chapter 15; ref. 3, chapter 5).

For cases where it is necessary to create input files directly from the keyboard, a brief description of the input data will be provided so the user will know what to expect. Stream sheet and blade geometry are input at a specified number of streamwise points. The point numbers corresponding to the leading and trailing edges must be specified. The number of computational streamwise points and the point numbers corresponding to the leading and trailing edges must also be specified. This is the only "tricky" part. The analyses use computational stations equal spaced in the meridional direction. This fact, together with the number of computational stations placed upstream and downstream of the blade passage dictate the meridional distances upstream and downstream of the blade passages where stream sheet data must be supplied. Users must be careful that their input data extends at least that far so that computational stations can be located by interpolation.

The blade angle convention setting technique discussed for Program FLOW3D is used here also. One set of splitter blades can be included. If splitters are included, the leading and trailing edge meridional locations must be specified as a fraction of the meridional length of the full-length blades. For the number of streamwise geometry input points specified, the axial coordinates, radial coordinates, stream sheet thickness, blade mean camberline angle and blade thickness are all supplied (blade data entered as zeros outside the blade passage). The blade angles are measured <u>in the</u>

stream surface. Inlet total pressure and total temperature are supplied. The blade speed and the inlet absolute tangential velocity are entered along with a radius at which they apply. This approach is needed since the upstream boundary of the computational domain is calculated rather than specified by the input data. The inlet mass flow (or discharge static pressure where needed for Program TDB2B, only) are supplied. The standard input/output units and gas equation of state packages are employed, but Program TDB2B requires use of the pseudo-perfect gas model only. That is an excellent model so long as the inlet conditions are not too close to the vapor saturation line. You will still have the convenience offered by the gas property database, but in the interest of practical computation speed, the full non-ideal gas model is not available.

For use in qualifying these analyses and to support program AIRFOIL, an alternate input style is included to handle axial-flow style blades. Basically, it involves entering (x, y) coordinates on the two blade surfaces and at the center of the blade leading and trailing edge "nose" radii, where x is measured along the blade chord and y is normal to it. A blade chord setting angle with the axial direction completes this specification. Should the user want to use this, this discussion plus the on-screen help available should be sufficient. The program also accepts the standard axial-flow compressor blades of ref. 3, chapter 4.

13.0 Program RIGPAC

Program RIGPAC (an acronym for Radial Impeller Geometry PACkage) is a general geometry package to compute and manipulate geometry for general vaned components. Basically, it performs functions for existing designs that the designer would obtain from Program GASPATH for new designs. As mentioned previously, this is often an entry point into the Turb*Aero* design and analysis system for an existing design. Some functions performed by Program RIGPAC are very difficult to perform manually from a basic drawing (e.g., throat area and blade angle calculations). This program can model the same four blade types as Program GASPATH, but the geometry can be entered in several different ways to better accommodate the specific data the user has available. Specifically, the blade geometry can be entered in the following forms:

- Cartesian coordinates (x, y, z) of points on the two blade surfaces.
- Cartesian coordinates (x, y, z) on the blade camberline and the blade thickness.
- Axial and radial coordinates (z, r), the camberline blade angle and the blade thickness. Note that the blade angle is defined in the surface of the contour being entered regardless of the blade type.
- Axial and radial coordinates (z, r), the camberline polar angle and the blade thickness.
- Axial and radial coordinates (z, r) and the blade thickness along with specifications for a 2D axial-element blade with a circular-arc camberline.

For 3D blades the data are specified on the hub and shroud surfaces. For 2D blades they are specified on the longer of these two surfaces (normally the hub) and only (z, r) coordinates for the other surface. There is an option to specify a thickness distribution on both hub and shroud for 2D blades if more flexibility is needed. From these basic specifications, the program inserts a specified number of equal area stream surfaces and computes all blade geometry on the stream surfaces that bound them.

Program RIGPAC also plays an important role in new designs. After the basic geometry is constructed, the Program will perform a number of "operations" to modify it without changing the original geometry data, including

- Impose a new shroud contour (which can be imported from a file previously exported from it or from Programs FLOW3D or GASPATH).
- Trim the blade tip diameter.
- Trim the blade inlet (leading edge).
- Scale all geometry.
- Impose a new shroud to change the gross area (the same function previously discussed for Program FLOW3D).
- Smooth an end-wall contour.
- Set blade trailing edge radii for 3D blades.
- Reverse the direction of rotation.
- Define inlet or discharge vaneless extensions.
- Rotate geometry to a new polar angle orientation.

Although not an "operation" a useful modification can be accomplished through the program's blade geometry edit capability. One can add an additional quasi-normal at the blade trailing edge, which will cause the program to extend the blade while maintaining a constant blade angle. In combination with the operation to change the passage gross area, this makes it very easy to add additional blade stock to the basic design, as is usually desired when ordering a casting pattern. The many modifications possible usually makes it useful to end the design process using this program. Program GASPATH can all create an input file for this program to simplify the process. After all operations are complete, Program RIGPAC can perform any of the following functions:

- Create another input file for itself with the final geometry using any of the available blade geometry input options (removes most of the operations and makes them permanent).
- Create or update component geometry data for Program FLOW3D input files.
- Export the final hub or shroud contour coordinates and blade data to files for subsequent importing into itself or into Programs FLOW3D and BEZIER.

14.0 Program RKMOD

Program RKMOD is a general idea/non-ideal fluid equation-of-state package (chapter 2 of refs. 1, 2 and 3 and ref. 4) for single-phase and mixtures of fluids using data from the gas property database. Simplified versions of this equation of state package are used in the various compressor design and analysis programs. Programs FLOW3D and TDB2B use a version restricted to the pseudo-perfect gas model (refs. 1, 2 and 3, chapter 2). In addition to its primary purpose of computing fluid properties, Program RKMOD provides valuable support to the overall *TurbAero* design and analysis system. It provides a means to evaluate the accuracy of gas property data and an equation of state by comparing gas property predictions against other data sources the user may choose to use as a "standard". To simplify this type of comparison, the program lets you assign reference point values of enthalpy and entropy for specified temperature and pressure to establish the same enthalpy-entropy reference as used by the "standard." It also lets you check to confirm the flow is free of liquid for the internal flow analysis programs (FLOW3D, B2B2D and TDB2B) which require gas-phase flow only. Three basic equation-of-state options are available:

- Ideal gas with temperature-dependent specific heats.
- The classical Redlich-Kwong non-ideal fluid equation of state.
- Aungier's modified Redlich-Kwong non-ideal fluid equation of state.

Program RKMOD performs the following specific functions:

- Creates tables of thermodynamic property data as a function of two specified thermodynamic parameters.
- Computes adiabatic head and efficiency from inlet and discharge conditions.
- Computes polytropic head and efficiency from inlet and discharge conditions.
- Computes discharge conditions from adiabatic head and efficiency and inlet conditions.
- Computes discharge conditions from polytropic head and efficiency and inlet conditions.
- Computes pseudo-perfect gas constants from specified total thermodynamic conditions and the corresponding sonic flow conditions (ref 1., section 2.7 and ref 2., section 2.9).
- Performs equilibrium flash (liquid knockout) calculations (ref 1., section 2.11)
- Creates tables of approximate liquid-phase property data (ref 1., section 2.10).
- Converts relative humidity to mole fraction of water to assist in forming humid gas mixtures.

Within the limits of the liquid-phase approximate models used, Program RKMOD will predict properties for gas-phase, liquid-phase and two-phase fluids. In principle, the non-ideal fluid equations of state can be applied directly to all fluid phases (indeed, this must be done for the equilibrium flash calculation). But, the approximate liquid phase models are a more reliable approach. And that approach is also consistent with the simplified equation-of-state packages used in other *TurbAero* design and analysis programs. So, if Program RKMOD indicates your problem is a valid gas-phase case, the other programs will be in agreement. Note that the viscosity model included is valid for gas-phase flow only.

15.0 Program GASDATA

Program GASDATA is a maintenance program for the gas thermodynamic property database file (RKDBASE.DAT). It performs the following basic functions:

- Lets you edit the property data in the input/output units you prefer.
- Lets you add additional component property data in the input/output units you prefer.
- Lets you form new components as mixtures of existing components.
- Lets you delete components from the data base.
- Lets you view or print the property data in the input/output units you prefer.
- Lets you rearrange the order of components in the data base (currently, they are in alphabetical order).
- Lets you save the data base under another file name (for back up).

File RKDBASE.DAT is an ASCII (text) file that can be viewed and modified in any text editor or word processor if you prefer. To do that, a few features need to be understood. First, line 1 of the file contains the number of components for which data are included in the database (currently over 100). For each component the property data consists of three lines of data entered as follows:

- Line 1: The component name, 20 character maximum, in quotation marks.

- Line 2: Five variables separated by a space or comma.

  - Molecular weight
  - Critical temperature (^{o}K).
  - Critical pressure (atm).
  - Critical compressibility factor.
  - Acentric factor.

- Line 3: Four constants (A, B, C, D), separated by a space or a comma, for a curve fit of specific heat at constant pressure (cal/gm/^{o}K) conforming to the following equation (T in ^{o}K).

$$c_p(T) = A + B(T/1000) + C(T/1000)^2 + D(T/1000)^3$$

16.0 Program EXHAUST

Program EXHAUST is an aerodynamic performance analysis for the turbine exhaust components (ref 1, chapter 12 and ref 3. chapter 13). It can predict the aerodynamic performance of exhaust diffusers, scrolls and collectors. Exhaust diffusers are modeled by a modified version of the general annular passage performance analysis for centrifugal compressors (ref. 2). Hence, nearly any type of diffuser can be analyzed, ranging from simple axial diffusers to curved (90° bend) type diffusers. The scroll and collector performance analysis is also based on methods of reference 2. The performance is quantified in terms of total pressure loss coefficients and static pressure recovery coefficients for all components analyzed and for the combination of all components.

The input data for the exhaust diffuser includes the mass flow rate, inlet thermodynamic conditions and the fraction of the passage width occupied by the end-wall boundary layers at the inlet. The root-mean-square surface finish and the number of equal-spaced stations along the passage that are to be used for the analysis (minimum of three) are specified. Then you supply geometry at a specified number of stations (minimum of two) from which the program will obtain the computation stations by interpolation. The data required are the axial coordinate and radius on the mean stream surface and the passage hub-to-shroud width. There is a special feature included that permits you to set the radius of any point other than end points to zero. This flags the station as one lying in a region where axial coordinate, radius and passage width all vary linearly with radius. The program will then compute intermediate stations using the end-point data for this section. The program has a fairly sophisticated data edit capability to make it easy to modify this data set. There is also a special option that models annular bends using two or three input stations and assumption that the angle of the mean stream surface with the axial direction varies linearly between the specified stations. Input for the optional scroll or

collector includes the root-mean-square surface finish, the passage area and mean radius at the station where the flow has been fully collected, and the discharge flange area. Various figures in chapter 13 illustrate the input geometry data required for all the components analyzed by program EXHAUST.

17.0 Program VOLUTE

Program VOLUTE is a detailed geometry design program for internal and external volutes for centrifugal compressors (ref.1, chapter 11) and radial in-flow turbines (ref. 4, chapter 10). It supplies detailed profile geometry for the passage contours at specified circumferential stations. It allows for use of either elliptical or rectangular passage cross-sections of specified aspect ratios. The picture to the right shows an external, elliptical volute. The user supplies the radius, R, passage width, b, and flow angle at the inner (exit) boundary. This is typically obtained from a performance analysis by program RIFT at the current phase of the stage design process. The passage cross-section areas are sized based on conservation of angular momentum. It is assumed that the gas density is constant throughout, which is normally a reasonable

A

B

Rc

R

b

External, Elliptical Volute

approximation. The passage areas can be undersized or oversized by specifying a multiplying factor usually referred to as the volute sizing parameter (a value of 1.0 corresponds to an ideal sizing that exactly conserves angular momentum. Ideally the volute area increases from zero to a maximum value at the entrance or full-collection station (where the passage contains 100% of the inlet flow). But a passage area of zero cannot be manufactured, so a minimum area constraint is imposed. This is based on the passage width, b, and requiring a reasonable curvature in the bend from the radial to the axial direction. The program supplies the key parameters of the ellipse or rectangle defining the passage contours at all circumferential stations. That is probably sufficient for most CAD software, but the program also supplies actual passage contour coordinates for all stations in case they are needed.
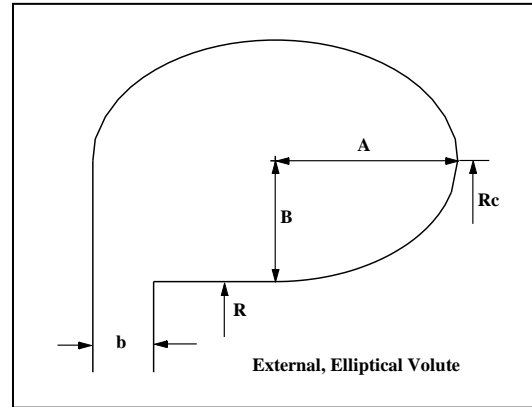
18.0 Program BLADE

Program BLADE was developed specifically to validate the data and logic needed to model and display the standard airfoil types for their use in the various axial-flow compressor programs in the *CompAero* system (ref. 3). It has been included in the *TurbAero* system because some users have found it useful for simple wind turbine blade design. It can design blade sections using any of the standard airfoil types. It can supply graphical displays and tabular blade surface and camberline coordinates for both isolated airfoil and cascade forms. It also calculates the throat spacing for the cascade configuration. These data can be viewed on the monitor screen or sent to the printer. The program can also export data to a text file in a format well suited for importing into spreadsheet or graphics software. This program is very simple to use and the on-screen help is quite sufficient to guide the user in all of its functions.

19.0 Importing and Exporting Design Data

Most programs have extensive data import and export capabilities. Import and export is in the form of ASCII text type format. The source or destination of the design data may be any application that can supply or use this type of data file. A common example is Microsoft Excel spreadsheets. For Excel, the space delimited text file (*.prn) is the appropriate file type to use when importing *TurbAero* data into Excel or in a "Save As" operation to provide Excel data for import into *TurbAero*. Export to the graphics application PSI-Plot by Poly Software International is another good example. It was the basis for most of the figures contained in reference 1, 2 and 3, using the (*.dat) file type expected by PSI-Plot.

The *TurbAero* general data export utility provides for a variety of column and line export styles that should support a wide variety of other application import capabilities. Extensive on-screen help is available for guidance, but you may need to experiment some to meet the requirements of other applications unless their documentation is clear on this point.

Programs BEZIER, GASPATH, FLOW3D and RIGPAC often exchange data to define end-wall contours via a

special import and export format handled internally by those programs. You can also create those data exchange files from other sources such as drawings, CAD or CAE applications, etc. using the following ASCII text format:

- On the first line, supply the number of coordinate points to be supplied to define the contour.
- Starting on the second line, enter all axial-coordinates. They can all be entered on one line or on several lines if you choose, but separate successive values on the same line with a blank space.
- Starting on a new line, supply all radial-coordinates in the same fashion.

Program RIGPAC has a number of special purpose export options (in addition to those described in section 13, above) that were incorporated over the years that may be beneficial to some users. These include exports for Pro/Engineer, the original B2B3D CFD code (by Professor Dawes). It also offers an impeller geometry export for the CAE/CAD application SolidWorks. This is a fairly generic geometry export which may be useful for other CAD/CAE/CFD applications. Users may want to check the online help for that export option to see if its content may be useful to support those types of applications.

Programs AXTURB and RIFT have a special option to export performance maps in files suitable for importing into Sinda/Fluint, a generalized fluid and thermal analysis for complex systems by C&R Technologies (http://www.crtech.com). Note that the sample problems (Section 20) contain performance maps that were generated before this capability was added. Hence, they lack some of the data needed for this export.

19.0 Equation of State Issues

Turbine aerodynamic design and analysis occasionally involves special issues relative to fluid thermodynamic modeling and the equation of state that are almost never encountered in compressors. Unlike compressors, turbines may operate very close to the vapor saturation line or even encounter two-phase flow. While the compression process tends to move downstream conditions away from the saturation line, the expansion process may do the opposite. This is a realistic condition that must be addressed in a performance analysis. Programs AXTURB, AFTSIZE, RIFT, RIFTSIZE and RKMOD use a rather complete equation of state model capable of operation with two-phase flow present. But the other design and analysis programs are limited to gas-phase flows only. In principle, the user need only avoid using those programs in the two-phase flow or "wet" region. Program RKMOD can list vapor saturation line data so you can check for this. But that may not be sufficient to avoid all problems. Many programs involve iterative numerical analysis to converge on an appropriate solution. Depending on the accuracy of the initial guess used to start the analysis, it is quite possible for the solution to drift into the wet region while the solution is far from convergence. If that occurs, the user may encounter a fatal error and have no indication of why it occurred. The usual cause is that program attempted a calculation using a specific volume close to zero, which raised a fatal error of some type.

This can be particularly troublesome when designing a component that operates very close to the vapor saturation line. Often inviscid internal flow analysis is needed for blade design using programs FLOW3D, B2B2D or TDB2B to evaluate blade loadings. These are all vulnerable to excursions into the wet region while the solution is far from convergence. Programs FLOW3D and TDB2B also employ the pseudo-perfect gas model in all cases. As pointed out in section 2.0, that model is often not appropriate for turbines, particularly for components operating with a non-ideal gas very close to the vapor saturation line. In those cases, the compressibility factor can change dramatically through the component, such that there is no pseudo-perfect gas model suitable to approximate the problem throughout the solution domain. The risk increases as the expansion increases, with analyses of components close to choke being particularly vulnerable. But in almost all cases a small change in state point conditions from conditions very close to the vapor saturation line usually results in a large change in compressibility factor.

There are steps you can take when internal flow analysis near the vapor saturation line is necessary. First, do not attempt to design your blades for operating conditions near choke. That is not good practice anyway, since blade loading predictions will not be very accurate or even very useful for those cases. Alternatively, do your internal flow evaluations for flow rates less than choke. But the most appropriate approach for cases close to the saturation line is to use the principle of similitude (ref. 1, chapters 1 and 8) to analyze an "equivalent" condition that is not too close to the vapor saturation line. Typically, Mach-number similarity is used to define a mass flow and speed that yields equivalent inlet

conditions for the revised state point. Reynolds number and volume ratio effects may compromise the equivalence obtained, but this may be your only option to accomplish a meaningful analysis.

Often engineers may employ a commercial viscous CFD code to address problems not adequately modeled by conventional aerodynamic analysis techniques. That approach will be of doubtful value in this case, since commercial CFD codes are very unlikely to offer thermodynamic models sufficient for this purpose. Indeed, most commercial CFD codes are very inferior to the TurbAero internal flow analysis codes in this respect.

20.0 Printed Output Issues and Alternatives

The various programs use a common output module that lets the user view or print tabular or graphical output. That module has performed flawlessly for Windows 3.1, 95, 98, ME and 2000. With the release of Windows XP, occasional problems have been encountered when printing the output files. No problems have been encountered in the more common practice of viewing them on the monitor screen. The problem is very unpredictable and apparently somewhat dependent on the printer in use. Early on it was discovered that simply changing the printer font size and then immediately changing it back (just so the code will recognize something was changed) just before printing would correct the problem (this can easily be done in the printed output window from the "Settings" menu). A fairly simple upgrade was incorporated which simply simulates that manual fix within the code. That upgrade resolved the problem on all of my PCs. But one report has been received of this problem occurring even after that upgrade was accomplished. The user should make note of the manual fix in case it is ever needed. Often the problem could be anticipated from the summary listed in the output window indicating a number of characters per line or a number of lines per page much greater than what was established during the printer setup.

There is also a useful alternative provided in the output window. The tabular output can be saved in a plain text file (typically with the file name extender .txt). This process removes the format codes used by the printed output module. These text files can be read directly by various document applications such as Microsoft Word, WordPad, Notepad, etc. Usually the default formatting used by those applications for plain text files is sufficient such that no conversion activity is required. The printed output is setup for a fixed pitch font (e.g., Courier or Courier New) with a font size and margins sufficient to permit at least eighty characters to be printed per line. Hence, the need to adjust margins, font or font size remains a possibility, although no case has been encountered so far where that was necessary. This capability was included as a convenient way to archive or transmit important output files electronically. But it also allows the use of the document application to print the tabular output files as an alternative should the user encounter the problem described in the previous paragraph.

21.0 Sample Problems

When the software is installed, sample input files for the various programs are installed in sub-directory SAMPLES. The sample problems are specific cases included in reference 1. Program RKMOD was not used in this process, so a representative sample input file SAMPLE.RK is also included.

References

1. Aungier, R. H., *Turbine Aerodynamics: Axial-Flow and Radial-inflow Turbine Aerodynamic Design and Analysis* (ASME Press, New York, 2006)
2. Aungier, R. H., *Centrifugal Compressors: A Strategy for Aerodynamic Design and Analysis* (ASME Press, New York, 2000)
3. Aungier, R. H., *Axial-Flow Compressors: A Strategy for Aerodynamic Design and Analysis* (ASME Press, New York, 2003)
4. Aungier, R. H., "Thermodynamic State Relations," *The Handbook of Fluid Dynamics* (R. W. Johnson, editor), (CRC Press LLC, Boca Raton, FL, 1998), pp. 4-29 - 4-34.
5. Ye, Z.-Q., "A Systematic Computational Design System for Turbine Cascades, Airfoil Geometry and Blade-to-Blade Analysis," *Trans. ASME, Journal of Engineering for Gas Turbines and Power*, July, 1984, pp. 598-605.