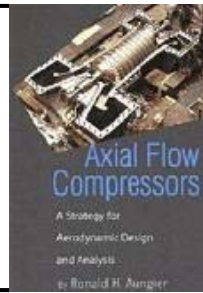


COMPAERO SOFTWARE USER'S GUIDE

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



The *CompAero* software implements the centrifugal and axial-flow compressor aerodynamic design and analysis strategy described in two books by R. H. Aungier (references 1 and 2). This description will make specific chapter and figure number references to those books to specifically relate the software to the descriptions they provide. 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 implemented and used successfully under the Microsoft Windows 95, 98, XP and Vista operating systems. This description is specific to version 1.50, released for sale on 6/20/09.

Contents

- 1.0 Software Installation and Execution
- 2.0 Features Common to Most Programs
- 3.0 Available Aerodynamic Design and Analysis Programs
- 4.0 Program SIZE
- 5.0 Program CENCOM
- 6.0 Programs GASPETH and BEZIER
- 7.0 Program FLOW3D
- 8.0 Programs VLDESIGN
- 9.0 Program VDDESIGN
- 10.0 Program RCDESIGN
- 11.0 Programs B2B2D and TDB2B
- 12.0 Program RIGPAC
- 13.0 Program RKM0D
- 14.0 Program GASDATA
- 15.0 Program AXCOM
- 16.0 Program AXSTAGE
- 17.0 Program AXDESIGN
- 18.0 Program EXHAUST
- 19.0 Program VOLUTE
- 20.0 Program BLADE
- 21.0 Program PTC10
- 22.0 Importing and Exporting Design Data
- 23.0 Printed Output Issues and Alternatives
- 24.0 Sample Problems
- References



A Typical Desktop Menu

1.0 Software Installation and Execution

Install the software by running program SETUP.EXE on the installation CD ROM or disk #1 of the installation disk set and follow the on-screen instructions. The installation disks contain compressed files, so they cannot just be copied to your hard drive. The program setup process will create a directory COMPAERO 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:\COMPAERO\SAMPLES) containing a set of sample input files for the various programs.

You can run all of the software by executing program COMPAERO.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 selecting a compact desktop menu (such as illustrated above). You can also run the programs individually using any of the usual Windows approaches, such as from shortcuts, a window or the Windows Programs menu. Program COMPAERO is usually a

more 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 the 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 operating system.

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 AXSTAGE, BEZIER, BLADE, GASPETH, RIGPAC and VOLUTE, (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 *CompAero* 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 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. Three basic options are available:

- Load Pseudo-Perfect Gas Data: Here you load the gas mole weight, specific heat, C_p , 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.
- 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 $C_p = C_p(T)$] 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 (as described in chapter 2). 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 will be adequate for nearly any centrifugal compressor application.

A good starting point for new users is to run Program RKM0D, 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 to COMPAERO.EXE the following computer programs are available for centrifugal and axial-flow compressor aerodynamic design and analysis.

- SIZE.EXE: a centrifugal compressor stage preliminary aerodynamic design program (ref. 1, chapter 6).
- CENCOM.EXE: an aerodynamic performance analysis for single-stage or multi-stage centrifugal compressors (ref. 1, chapter 5).
- GASPETH.EXE: a general gas path (end-wall contours and blades) design program (ref. 1, chapter 7).
- VLDESIGN.EXE: a vaneless diffuser detailed aerodynamic design program (ref. 1, chapter 8). Includes a performance analysis (ref. 1, chapter 5).
- VDDESIGN.EXE: a vaned diffuser detailed aerodynamic design program (ref. 1, chapter 9). Includes a performance analysis (ref. 1, chapter 5) and a linearized blade-to-blade flow analysis (ref. 1, chapter 12).
- RCDESIGN.EXE: a crossover bend and return channel detailed aerodynamic design program (ref. 1, chapter 10). Includes a performance analysis (ref. 1, chapter 5) and a linearized blade-to-blade flow analysis (ref. 1, chapter 12).
- FLOW3D.EXE: a quasi-three-dimensional inviscid flow field analysis (ref. 1, chapter 12 and ref. 2, chapter 12) with supporting surface boundary layer analyses (ref. 1, chapter 15 and ref. 2, chapter 5).
- B2B2D.EXE: a two-dimensional blade-to-blade potential flow analysis (chapter 13) with supporting surface boundary layer analyses (ref. 1, chapter 15 and ref. 2, chapter 5).
- TDB2B.EXE: a time-marching blade-to-blade flow analysis (ref. 1, chapter 14 and ref. 2, chapter 5) with supporting surface boundary layer analyses (ref. 1, chapter 15 and ref. 2, chapter 5).
- RKMODO.EXE: an ideal/non-ideal fluid equation-of-state package for a range of thermodynamic property calculations (refs. 1 and 2, 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 RKMODO and the other programs that use its equation of state.
- RIGPAC.EXE: (Radial Impeller Geometry PACKage): a general geometry package primarily for 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 GASPETH, 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 7 of ref. 1.
- BEZIER.EXE: a program to generate the curves used by Program GASPETH to construct the end-wall contours and blades (ref. 1, chapter 7). It can be accessed directly from Program GASPETH. But, on occasion it is convenient to work with this program for creating or refining curves just to avoid the need to navigate through the menu structure of Program GASPETH to get to this sub-program.
- AXCOM.EXE: an aerodynamic performance analysis for axial-flow compressors (ref. 2, chapters 7- 9).
- AXSTAGE.EXE: an axial-flow compressor stage design program (ref. 2, chapter 10).
- AXDESIGN.EXE: a multistage axial-flow compressor design program (ref. 2, chapter 11).
- EXHAUST.EXE: an axial-flow turbomachinery exhaust diffuser, scroll, collector aerodynamic performance analysis (ref. 2, 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 standard airfoils used (ref. 2, chapter 4). Its original purpose was to check out the blade geometry routines and database for axial-flow compressor. Sometimes it has been useful for generating blade section geometry for documentation purposes, so it has been included. Indeed, many of the figures included in reference 2 were generated using data exported from this program.
- PTC10.EXE: a program to correct test performance data on a centrifugal compressor to the intended working fluid and operating conditions appropriate to the intended application.

File RKDBASE.DAT, contains gas thermodynamic data for over 100 compounds collected over several years. It is supplied as a courtesy to the user, who must use the data at his or her own risk. No general statement is possible regarding either its accuracy or its adequacy for the user's range of thermodynamic conditions. Indeed, one function of Program RKMODO 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 SIZE

Program SIZE 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" centrifugal compressor 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 SIZE creates the input data files for other centrifugal compressor aerodynamics programs needed for the detailed stage design, including:

- Program CENCOM for the aerodynamic performance analysis.
- Program GASPETH for detailed design of the impeller hub, shroud and blades.
- Program VLDESIGN for detailed design of the vaneless diffuser.
- Program VDDESIGN for detailed design of the vaned diffuser.
- Program RCDESIGN for detailed design of the crossover/return channel.

Some editing of the input files created may necessary. Scaled blade thicknesses and recommended surface finish values may not be appropriate for manufacturing or mechanical design considerations. The component inlet flow data for the component design programs (VLDESIGN, VDDESIGN & RCDESIGN) are approximate. Usually, these should be revised using Program CENCOM predictions. Program CENCOM can perform these input file updates for you when it has completed a performance analysis.

To start your program SIZE run, you specify either adiabatic or polytropic as the performance data type to be used. You define the stage component types to be used.

- Impeller: covered or open; full-inducer, semi-inducer or no-inducer.
- Diffuser: vaned or vaneless.
- Discharge: return system or volute.

Full-inducer and semi-inducer impellers use straight-line element three-dimensional blades, while no-inducer designs use two-dimensional, axial-element, circular-arc blades. You specify the inlet total pressure and temperature for the stage and the lower and upper diameter limits for the stage (i.e., limits imposed by the shaft plus minimum hub metal thickness and by the casing size). If no limits are specified, the program chooses values without constraint. You have to define the input/output units you are using and the gas equation of state to be used.

The stage sizing calculations require specifications of rotation speed, impeller diameter, mass flow, head, efficiency and impeller tip flow angle. You have several options for providing these specifications.

Speed/Diameter Input: specify two of the following:

- Rotation speed (rpm)
- Impeller tip diameter
- Impeller tip speed, rotational Mach number or (standard air) equivalent tip speed.

The rotational Mach number is the tip speed divided by the inlet total sound speed. The equivalent tip speed option defines a tip speed that gives the same rotational Mach number as the specified value for standard air.

Inlet flow input options: specify the inlet flow coefficient, ϕ , inlet mass flow or inlet volume flow (Q). For impeller tip diameter, d_2 , and tip speed U_2 ,

$$\phi = Q / [\pi U_2 d_2^2 / 4]$$

The impeller tip triangle specification is either the tip relative flow angle or tip flow coefficient (tip meridional velocity / tip speed). The program makes a preliminary selection based upon diffuser type and inlet flow coefficient. The program displays this choice for you to edit as you choose.

The program offers an estimate of the target head coefficient and efficiency considered reasonable design targets if constraints permit (see ref 1., figures 6-1 and 6-2). You may use these values or enter your own choices. When you specify the head/efficiency targets, you enter the efficiency and one of the following: head coefficient, head, power, discharge total pressure or total-to-total stage pressure ratio. Use some care when not entering head coefficient. You can ask for values of the other parameters that are impossible for the diameter and speed you are using. Usually it is

best to use head coefficient until you have defined design data capable of producing the other design parameters before using them directly.

The stage sizing calculations assume the head/efficiency targets are accurate. When in doubt about this, it is advisable to run Program CENCOM following the sizing calculations (Program SIZE will create the input file) and revise these specifications for a rerun if necessary.

All Program SIZE input specifications can be edited interactively to refine the sizing calculations. After each component is sized, you are given the option to continue or to go back, edit the basic design data and start the sizing calculations from the beginning. You also can size only part of the stage components if you choose.

With this brief introduction and the fairly extensive on-screen help available from Program SIZE, it is really a fairly easy program to learn to use.

5.0 Program CENCOM

Program CENCOM provides an aerodynamic performance analysis for single-stage or multi-stage centrifugal compressors using the procedures described in ref. 1, chapter 5. For new designs, the initial input file for this program will normally be supplied by Program SIZE. 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. Only for existing compressor stages not designed with the *CompAero* system, will users have to enter all input data. But, it will often be necessary to edit input data, so a brief discussion of the data required will be given here.

Program CENCOM plays a very central role in the design process. Hence, it includes a capability to update input files for Programs VLDESIGN, VDDESIGN and RCDESIGN to supply them with current inlet fluid dynamic and thermodynamic data. It can also supply impeller flow data to FLOW3D and GASPAC. While Program SIZE can create initial input data files for most component detailed design programs, as components are refined, inlet conditions for downstream components will change. Program CENCOM will update that portion of the various input files for you. In the case of Programs FLOW3D, this is an easy way to add inlet flow data and the gas thermodynamic equation of state to an impeller input file created by Programs GASPAC or RIGPAC.

This program can collect and save performance prediction results and use them to display or print performance maps for the compressor. Three maps are provided: a pressure or head 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 speeds or guide vane settings and plot as separate curves) and let you transfer to the Map window to load or edit data. 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 predictions in the map file. On-screen help describes other features you need to know. The program always provides stage performance maps when a stage is analyzed. There are several advantages of this more general map capability. First, it maps the complete compressor performance rather than just a stage. Second, you can supply test data (if available) to assist in evaluating the program's accuracy or target performance goals for guidance in design activity. Finally, you can include results from several analyses, whereas a single analysis allows a maximum of eight flow points.

Input data for this program is entered for a series of stage components, which you can arrange in the order in which they appear in the stage. Each time you add a component, it is placed at the end of the input file. The program provides a fairly complete editing capability that, among other things, lets you easily rearrange the order of the components when appropriate. Some "components" are not actual stage components (e.g., units, gas model and inlet flow data) and they can appear in any order and at any location among the true components. When editing specific input data, you will simply choose the "component" to be edited and the program lets you change values of input parameters for it. Multi-stage compressors are analyzed by simply combining input files of the various stages to be included. The program lets you choose these stage input files and combines them into a multi-stage input file for you. When entering or editing input data, Program CENCOM offers extensive on-screen help

Inlet Flow And Rotation Speed Input Data

You enter or edit the inlet flow, total pressure, total temperature, swirl parameter and rotation speed for your case. The rotation speed is constant for each case, but up to eight flow points can be considered for each case. You have alternate options for the input for three of the parameters that you can select. The choices are:

- Inlet Flow: mass flow, volume flow or flow coefficient, ϕ . Use the last option only if an impeller is one of the stage components.
- Swirl Parameter: absolute tangential velocity or absolute flow angle with respect to tangent, on the mean streamline at the inlet of the first component.
- Rotation Speed: rpm, rotational Mach number (tip speed/inlet total sound speed), equivalent tip speed (sets actual tip speed to yield the same rotational Mach number as would occur for air at standard conditions at this equivalent tip speed) or the actual (mechanical) tip speed. Rotational Mach number and equivalent tip speed are converted to rpm using the inlet total sound speed for the first flow point. Hence, they are normally useful only when inlet total conditions are constant for all flow points.

Dimensional data are all entered in the input/output units you are using for this case. The program continually reorders your data such that the inlet flow is in increasing order. This is done every time you finish editing an inlet flow value. Several useful data entry and edit functions are available and are described in the on-screen help for this window. In the case of multistage compressor analyses, program versions dated 10/19/07 or later allow the user the option of specifying different rotation speeds for stages after the first to treat integral-gear compressors with stages mounted on separate pinions. Guidance is available in the on-screen help for this window.

Inlet Passage Input Data

The inlet passage component is just a starting point for the analysis where you can specify the inlet flow data when the first component is not an inlet guide vane or an impeller. The program will perform a mass balance on this annular passage station to start the performance analysis. Data required are the mean stream surface radius and the passage hub-to-shroud width.

Inlet Guide Vane Input Data

The geometry required for inlet guide vanes is as follows:

- The vane inlet mean stream surface radius.
- The vane inlet mean camberline angle with respect to the tangential direction.
- The vane inlet passage area (full area of the annulus).
- The vane discharge mean stream surface radius.
- The vane discharge mean camberline angle with respect to the tangential direction.
- The vane discharge passage area (full area of the annulus).
- The number of vanes.
- The vane meridional length, i.e., the distance from leading edge to trailing edge measured in the meridional (constant polar angle) plane.
- The vane mean angle, i.e., the vane mean camberline angle with respect to the tangential direction mid-way through the vane passage.

Note that all vane angles are with the vane set in its operating condition (for adjustable vanes, all angles must be adjusted when the vane is reset).

Impeller Geometry Input Data

Much of the impeller geometry is related to the mean stream surface (figure 5-4). This is a surface that divides the annulus into two equal areas. Typically, it is defined by the radius at the root-mean-square (rms) of the hub and shroud radii. Blade angles are measured in the relevant stream surface with respect to the tangential direction (figure 5-5). Values are input for the hub and shroud stream surfaces at the blade leading edge. Values required on the mean stream surface are at the leading edge, trailing edge and a mid-passage location.

Diameters for the blade leading and trailing edges at both hub and shroud are required. The disk and cover diameters (outer) are input. For open impellers, enter the cover diameter as zero. The eye diameter is an optional input. If not supplied, impeller thrust calculations will not be provided. You will also put in a tip diameter. Usually, this is

redundant, but is provided for cases where then hub and or shroud extend beyond the blade trailing edge. It is an average of the hub and shroud values.

Passage widths relevant to the blade leading and trailing edges are required. They should be representative of the passage width available for the flow. They are measured along quasi-normals (lines approximately normal to the hub and shroud - they are not necessarily aligned with the blade edge). The passage width at the eye is optional (again needed to provide impeller thrust). A passage width at the tip is also supplied. Usually, this is redundant, but is provided for cases where then hub and or shroud extend beyond the blade trailing edge.

Blade thicknesses at the leading and trailing edges on the mean stream surface are input. If more convenient, use average of hub and shroud values. Fillet blockages for the leading and trailing edges are also entered. These define the fraction of otherwise open area blocked by fillet welds (or any other area blockage not attributable to the blades, themselves). The mean stream surface cone angle, α_c , is the angle between a tangent to the mean stream surface and the axial direction (figure 5-4). Values are required at the leading and trailing edges. Surface finishes (rms) are required for both the internal passages and the hub/cover/housing area. Clearances between the housing and both the disk and cover are entered. In the case of open impellers, the blade-shroud clearance replaces the cover clearance. Use average values. You enter both the number of full blades and the number of splitter blades and the meridional length of each on the mean stream surface (e.g., L on figure 5-4; enter zeros all for splitter blade data if there are none).

Throat area is a required input (figure 5-5). You do have an alternative -- you can input the blade angle and blade thickness at the throat (on suction surface side of the blade passage) and the program will try to estimate it. This is not highly recommended, but may be okay in early phases of design and analysis or when choking is not expected in the impeller. Make one choice or the other and enter zero(s) for data not being entered. The best source of this data is to let one of the geometry programs (GASPATH or RIGPAC) create or update the input file for this program so an accurate throat area is obtained. You also select an inlet loss model to be used. This approximates the loss between the impeller eye and the blade leading edge. If you are modeling this loss separately (e.g., via a vaneless passage analysis) choose the option for no loss. The other choices are an inlet passage loss (typically for multi-stage configurations) or a nose-spinner loss (typical of axial inlet, overhung shaft).

As optional input, you can enter the curvatures (inverse of the radius of curvature) of the mean stream surface at the leading and trailing edges. If you do not enter them, the program will estimate them from the other data. Unless you have a good source for these data, it is best to let the program estimate them. They directly impose the hub-to-shroud meridional velocity gradients due to passage curvature. These curvatures are positive in the conventional impeller (surface concave upward). If you have a good source (e.g., detailed geometry design/analysis programs) supply the data since the program hasn't enough data to estimate them rigorously. But, the program's estimate is much better than a guess. When programs GASPATH and RIGPAC create or update impeller geometry for this program, they will supply these data.

Labyrinth seal geometry is supplied for the shaft seal. For covered impellers, the cover (or eye) seal geometry is also supplied. Seal leakage can be neglected for either or both seals by setting the number of seal fins to zero. If the seal leakage is neglected, the program will not provide an impeller thrust force calculation, since leakage greatly affects the pressure distributions along the disk and cover. Only straight-through labyrinth seals are modeled. The geometry input data common to both seals are (figure 4-6):

- Seal clearance: the clearance between the tip of the fins and the seal housing.
- Point thickness: the thickness of the seal fins at the tip or point.
- Seal pitch: the distance between adjacent fins.
- Number of fins: the number of fins in the seal.
- Seal diameter: the diameter at the clearance gap.

The shaft seal uses two more parameters to permit assigning the seal pressure and temperature at the outer side of the seal. If the stage has a return channel, these are only first guesses - the program will iterate to use the correct values. The input form is chosen to permit reasonable approximations for most shaft seals. The input data are:

- $(T_{\text{seal}} - T_{\text{oin}})/(T_{\text{tip}} - T_{\text{oin}})$
- $(P_{\text{seal}} - P_{\text{oin}})/(P_{\text{tip}} - P_{\text{oin}})$

where T_{seal} is the seal temperature, P_{seal} is the seal pressure, T_{oin} is the stage inlet total temperature, P_{oin} is the stage inlet

total pressure, T_{ip} is the impeller tip temperature, and P_{tip} is the impeller tip pressure. For a single stage compressor with outer seal exposed to inlet conditions, these data would both be entered as zero. For return channel stages, the temperature term would be about 1.0. The pressure term is typically about 1.3 - 1.4 (the program will correct both for return channel stages). For other cases the user must decide on appropriate values.

Vaneless Diffuser/Vaneless Passage Input Data

The input data for vaneless diffusers and vaneless passages are identical (a crossover bend can also use this input form, but usually the special form discussed later in this section is more convenient). You specify the rms surface finish and the number of equal spaced stations along the passage that are to be used for the analysis (minimum of three). 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.

Vaned Diffuser Input Data

The user should carefully note the discussion in section 5.5 regarding the types of vaned diffusers that can be analyzed by Program CENCOM. Most of the input data required are shown on figure 5-9, including r_2 , b_2 , r_3 , b_3 , β_3 , r_4 , b_4 , β_4 and h_{th} . Note that stations 3 and 4 are inside the vanes (e.g., at the center of the vane leading and trailing edge "nose" radii). Additional input data required include the number of vanes, the rms surface finish, leading and trailing edge vane thicknesses and the vane mean camberline angle (with respect to tangent) at a location mid-way along the vane.

Crossover Bend Input Data

When loading data for a crossover bend, you will have the option to use the general vaneless passage style input discussed above or the simpler method described here. The crossover hub and shroud contours can be approximated by elliptical contours. You supply the inlet radius (discharge radius assumed to be identical), the inlet and discharge passage hub-to-shroud widths, the axial axis of the shroud ellipse (i.e., the total crossover axial length), the radial semi-axes for both ellipses, the number of streamwise computing stations to be used and the rms surface finish. In some cases, the return channel vane leading edge may lie at a lower radius than set for the crossover discharge by this simple input style. If the difference is small, it can be ignored since the program will start the return channel analysis by imposing conservation of angular momentum from the crossover exit conditions and perform a mass balance at the return channel inlet. If the difference in radius is significant, a vaneless passage can be inserted between the crossover bend and the return channel.

Return Channel Input Data

Data required for the return channel are as follows:

- The blade inlet radius inside the blade (e.g., at the center of the leading edge "nose" radius).
- The blade inlet passage hub-to-shroud width approximately normal to the flow.
- The inlet blade mean camberline angle with respect to the tangential direction.
- The inlet blade thickness (e.g., twice the leading edge radius).
- The blade discharge radius inside the blade (e.g., at the center of the trailing edge "nose" radius).
- The blade discharge passage hub-to-shroud width, approximately normal to the flow.
- The discharge blade mean camberline angle with respect to the tangential direction.
- The discharge blade thickness (e.g., twice the trailing edge radius).
- The number of vanes in the return channel.
- The throat width, i.e., minimum distance between adjacent vanes.

- Mean camberline angle with respect to tangent at about mid-way through the blade passage.
- The root-mean-square (rms) surface finish.
- The angle between mean stream surface and the axial direction at the blade leading edge.
- The angle between mean stream surface and the axial direction at the blade trailing edge.
- The angle between mean stream surface and the axial direction at the end of the exit turn (normally zero, set to blade trailing edge value to omit exit turn loss, e.g., if you want to treat it with a separate vaneless passage analysis).

Volute (Scroll) And Collector Input Data

Volutes (scrolls) and collectors use the same input data (but have quite different performance). The data required defines the full collection plane, which is the cross-section in the volute or collector where all of the flow from the diffuser has been collected and is ready to exit into the exit cone or discharge pipe. The input data required are (figure 5-14):

- The full-collection plane cross-sectional area, A_6 .
- The full-collection plane mean radius, r_6 .
- The area at the discharge of the exit cone, A_7 .
- The root-mean-square (rms) surface finish.

If the $A_7 < A_6$, the exit cone portion of the loss model, equation (5-110), will be set to zero, i.e., the exit cone loss is neglected.

Total Pressure Loss Input Data

A total pressure loss can be imposed at one or more locations within the stage. This is to permit you to account for a loss source not modeled by the program specifically. Probably the most common use is to impose an exhaust loss at the end of the stage. The data required are a meridional loss coefficient, ϖ_m , a tangential loss coefficient, ϖ_u , and a total pressure drop, Δp_t to impose the total pressure loss as defined in equation (5-111). Usually, Δp_t is set to zero, but it is included for added flexibility if required. So, if you want to convert the stage discharge performance data from total to static data, simply impose an exhaust loss after the last component with ϖ_m and ϖ_u set to 1.0 and Δp_t set to 0. This is the only "component" you can rename for clarity, e.g., the previous example might be renamed to "Exhaust Loss". This loss can appear as many times in the stage as you wish. When it follows a scroll or collector, note that the program sets $C_U = 0$ after those components, i.e., C_m is used, since it is a "through-flow" velocity (even though it looks like a tangential velocity).

Liquid Knockout

Liquid knockout will perform an equilibrium flash calculation (chapter 2) at the local total temperature and total pressure and drop any liquid from the gas stream. This is typically used only after a cooler, but can be invoked anywhere. To invoke the liquid knockout, you must be using one of the non-ideal gas models and the gas must contain more than one component -- otherwise, the run is aborted with an error message. There is no input data required -- you simply insert this fictitious "component" in the location where the liquid knockout calculation is to be performed. The program will update the mixture equation of state and the mass flow rate to account for the liquid dropped from the gas stream.

Inter-Cooler Input Data

An inter-cooler (heat exchanger) can be included as the first component in the stage. It imposes a total pressure loss and a total temperature drop on the fluid inlet conditions before starting the stage performance analysis. It is useful when dealing with a downstream section of stages in a multi-stage machine. The performance (efficiency, etc.) is based on conditions out of the inter-cooler. You specify a flow area to be used for imposing the total pressure loss defined by equation (5-112), the loss coefficient, Δp_t and the constants A and B in equation (5-113). The program performs a mass

balance on that area to obtain the gas density and velocity used in equation (5-112). If either the flow area or the loss coefficient is entered as zero, no total pressure loss is applied.

Extraction Flow Input Data

An Extraction Flow component removes part of the mass flow from the compressor main flow, typically at the end of a return channel. The program makes this adjustment, and defines the flow properties for the adjusted mass flow. The extraction flow is not used in computing stage performance data (efficiencies, etc.), but new exit conditions for a following stage are computed.

You can specify either the fractional extraction mass flows (ratio of extraction mass flow-to-upstream mass flow) or the discharge flange static pressure (entered as a negative number). Enter zero and the saturated vapor pressure is used as the flange static pressure (this is always used as the minimum flange pressure). You enter flow/flange pressure data for the eight (possible) flow points. A value of zero on points after the first will be replaced by the first point's value (so, if you really want zero, all points must be zero).

Other data required are (figure 5-21) the compressor passage area after the extraction and its mean radius, the discharge flange area, the extraction loss coefficient and an area associated with it to be used in equation (5-114). If you specify extraction mass flow, the flange calculation can be omitted by entering the area for the loss coefficient and/or the flange area as zero.

Side-Load Flow Input Data

A side-load flow adds mass flow to the stage by joining an external side stream flow with the stage flow. A side load flow can only appear at the end of the stage (typically, following the return channel). The side load flow is not used in computing stage performance data (efficiencies, etc.), but new exit conditions for a following stage are computed.

You specify either the ratio of the side load mass flow to the stage mass flow or the flange static pressure (entered as a negative number). Enter zero to use the vapor saturation pressure as the flange static pressure (this is always used as the minimum flange static pressure). You enter flow/flange pressure data for the eight (possible) flow points. A value of zero on points after the first will be replaced by the first point's value (so, if you really want zero, all points must be zero).

Other data required (figure 5-20) are the (external) flange total temperature, the stage and side load passage areas where the two flows merge, the passage area and mean passage radius after merging, the inlet flange area, a side load loss coefficient and an area associated with it for use in equation (5-114). The area for the loss coefficient is used to compute a gas density and velocity by a mass balance for use in equation (5-114).

6.0 Programs GASPETH and BEZIER

Program GASPETH (ref. 1, 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 GASPETH, 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 GASPETH'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 must be the same as that of the gas path geometry file, and its file name extender must be .BEZ so that Program GASPETH can find it. When the curve file is generated through Program GASPETH 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 impeller design, but it can be used for almost any component. Program SIZE is normally used to create its input files. It has provision for creating or updating geometry input files for Programs CENCOM 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 GASPETH 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. If you create a gas data input file for this program using Program

CENCOM, an approximate hub-shroud and blade loading analysis is available for guidance. This is not as accurate as a Program FLOW3D analysis, but can be convenient for easy reference.

Becoming familiar with Programs GASPETH 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 user will find that the programs are actually quite easy to use.

The program can design four 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, equations (7-1) provide 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 SIZE 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 if you delete curves (hence, changing numbers), you must reselect curves. 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 will 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 the 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 offers more flexibility in thickness distributions on 2D blades.

7.0 Program FLOW3D

Program FLOW3D provides a quasi-three-dimensional internal flow analysis and supporting end-wall and blade surface boundary layer calculations (refs. 1 and 2, chapter 12). Program FLOW3D is an integral part of the centrifugal compressor design and analysis system of reference 1, where its most common use is in impeller design. In that role, it is a primary evaluation tool for the gas path design generated by Program GASPETH. When used in impeller design, its geometry input file data will normally be generated (and updated) by Program GASPETH. Program CENCOM 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 centrifugal compressors. Its most common use for axial-flow compressors is to conduct a more detailed evaluation of the internal flow in blade passages where the designer has special concerns. Program AXCOM and AXDESIGN have the capability to supply a complete input file for an analysis with this program. So, users will rarely need to create or edit input files for this program. 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 centrifugal compressors. The fourth is quite general, but also quite difficult to define without a separate blade design system, but certainly possible. Programs GASPETH

and RIGPAC provide that capability for 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. 2, 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 *CompAero* design and analysis system. In many cases, you may find it more convenient to model centrifugal compressor components in Program RIGPAC, which accepts a wide range of geometry input options and basically performs all of the functions of Program GASPETH for existing geometry. Program RIGPAC can set up an input file for this program and component geometry for Program CENCOM to give you a simpler entry into the *CompAero* 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 GASPETH 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 ratios 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 GASPETH 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 know 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 GASPETH 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 in the stream surfaces defined by those contours. 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 in the surface defined by that coordinate. 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 only the pseudo-perfect gas model. 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

8.0 Program VLDESIGN

Program VLDESIGN performs the detailed aerodynamic design of vaneless diffusers. Ref. 1, chapter 8 describes the approach used in detail and is sufficiently brief to leave little need for further discussion of details and input data. Usually, Program SIZE generates its first input file, with Program CENCOM providing updated inlet aerodynamic data after the impeller design is finalized. Note that due to the provision for mixed-flow stages (ref.1, figure 8-1), Program CENCOM hasn't sufficient data to update the impeller tip geometry. So, if that data is modified in the detailed impeller design process, be sure to edit the data supplied by Program SIZE. When a candidate detailed design is generated, Program VLDESIGN can update the vaneless diffuser data in Program CENCOM files.

9.0 Program VDDESIGN

Program VDDESIGN performs the detailed aerodynamic design of convention, thin vaned or airfoil style vaned diffusers. Ref. 1, chapter 9 describes the approach used in detail and is sufficiently brief to leave little need for further discussion of details and input data. Usually, Program SIZE generates its first input file, with Program CENCOM providing updated inlet aerodynamic data after the impeller design is finalized. Program CENCOM doesn't have sufficient data to reliably update the inlet geometry. So, if that data is modified in the detailed impeller or vaneless diffuser design process, be sure to edit the data supplied by Program SIZE. When a candidate detailed design is generated, Program VDDESIGN can update the vaned diffuser and upstream vaneless space geometry data in Program CENCOM files. It can also create an input file for Program RIGPAC

Program VDDESIGN will give the option to either round the blade trailing edge or cut it off on a constant radius to conform to the desired manufacturing process. It will perform design point or off-design performance analyses (ref. 1, chapter 5) to guide the design process without continually returning to Program CENCOM for that purpose. It will also perform a linearized blade-to-blade flow analysis (ref. 1, chapter 12) for direct evaluation of the blade loading. Its line printer output includes a detailed description of the contour and vane geometry designed.

10.0 Program RCDESIGN

Program RCDESIGN performs the detailed aerodynamic design of the crossover and return channel. Ref. 1, chapter 10 describes the approach used in detail and is sufficiently brief to leave little need for further discussion of details and input data. Usually, Program SIZE generates its first input file, with Program CENCOM providing updated crossover inlet aerodynamic and geometric data after the diffuser design is finalized. When a candidate detailed design is generated, Program RCDESIGN can update the crossover and return channel geometry data in Program CENCOM files. It can also create an input file for Program RIGPAC

Program RCDESIGN will perform design point performance analyses (ref. 1, chapter 5) to guide the design process without continually returning to Program CENCOM for that purpose. It will also perform a linearized blade-to-blade flow analysis (ref. 1, chapter 12) for direct evaluation of the blade loading. Its line printer output includes a detailed description of the contour and vane geometry designed.

11.0 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 and ref. 2, chapter 5) suitable

for subsonic and slightly transonic Mach number levels. Program TDB2B is a general time-marching method (ref. 1, chapter 14 and ref. 2, 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 been given provision to create a restart file from which an analysis that has been terminated can be restarted from the point where it left off.

The most common use of the two-dimensional programs is a final blade loading evaluation for impellers to be sure the linearized method in Program FLOW3D achieved sufficient accuracy. In this role, its input file is created by Program FLOW3D. All three programs offer a blade surface boundary layer analysis (ref 1., chapter 15), including the profile loss coefficient model of ref 1, equation (15-35). Programs AXCOM and AXDESIGN can supply a complete input file for this analysis.

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 in the 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 these analyses will use only the pseudo-perfect gas model. 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, 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 standard airfoil types of ref. 2, chapter 4.

12.0 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 GSPATH for new designs. As mentioned previously, this is often the best entry point into the *CompAero* 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 GSPATH, 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 leading 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. Programs GASPATH, VDDESIGN and RCDESIGN 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 CENCOM input files.
- 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.

13.0 Program RKM0D

Program RKM0D is a general idea/non-ideal fluid equation-of-state package (chapter 2 of refs. 1 and 2 and ref. 3) 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 SIZE, CENCOM, VLDESIGN, VDDESIGN, RCDESIGN, AXCOM AND AXDESIGN use a form restricted to the gas-phase. Programs FLOW3D, B2B2D and TDB2B use a version restricted to the pseudo-perfect gas model (Ref. 1, chapter 2, section 2.7). In addition to its primary purpose of computing fluid properties, Program RKM0D provides valuable support to the overall *CompAero* design and analysis system. First, 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". Secondly, it provides a means to check on compressor component operating conditions to be sure that the fluid is in the gas phase. Since none of the other *CompAero* design and analysis programs can be used in the

"wet" region, there are times when the user may want to check this. 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 RKM0D 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 water mole fraction to assist in forming humid gas mixtures.

Within the limits of the liquid-phase approximate models used, Program RKM0D 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, this writer finds the approximate liquid phase models to be a more reliable approach. And, more important, that approach is consistent with the simplified equation-of-state packages used in other *CompAero* design and analysis programs. So, if Program RKM0D indicates your problem is a valid gas-phase case, the other programs will be in agreement. Due to an oversight, the gas viscosity model is not described in ref. 1, but can be found in ref. 2.

14.0 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 ($^{\circ}\text{K}$).
 - 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/°K) conforming to the following equation (T in °K).

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

15.0 Program AXCOM

Program AXCOM is a general aerodynamic performance analysis for axial-flow compressors using the analysis procedures described in ref. 2, chapters 6 - 9. You can enter all input data to treat an existing compressor design. When program AXDESIGN is used to design a compressor, it can create a complete input file for use by program AXCOM for off-design performance analysis. Program AXCOM can also extract blade design data from an input file for program AXSTAGE when that program has been used to design a compressor stage. After a performance analysis has been run, program AXCOM can create input files for programs B2B2D and TDB2B for selected blade rows and stream surfaces and for program FLOW3D for selected blade rows. That can be useful to obtain an internal flow analysis of blade sections or blade rows when you have concerns about their performance (e.g., if Mach numbers become super-critical, you might want to confirm that internal Mach numbers are not excessive).

Program AXCOM can analyze performance for known end-wall contours, or it can calculate one of the contours by sizing the annulus as described in ref. 2, chapter 7. In the latter case, you can also smooth the calculated contour one or more times before switching to the more accurate fixed-contour option. Annulus sizing is the approach used to apply a standard stage design as described in ref. 2, chapter 10. Annulus sizing neglects stream surface curvature effects. Fixed-contour solutions can use any of the normal equilibrium models described in ref. 2, chapter 7.

This program can collect and save performance prediction results and use them to display or print performance maps for the compressor. Two maps are provided, one a pressure or head type and the other an efficiency type. These maps can include experimental performance data if available. The map menu in the main window lets you select the variables to map, the curve legend (e.g., to identify alternate speeds or guide vane settings and plot as separate curves) and let you transfer to the Map window to load or edit data. 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 predictions in the map file. On-screen help describes other features you need to know.

The program continually checks your input data and provides a status summary on the screen to advise you of any missing or inconsistent data. It offers a very wide range of data options for viewing the input and calculated data, including both text and graphical presentations. The same data can be inserted into an output file for printing. The program warns you if any data has been added or changed but not saved. The program has a data export capability to save data in text files in a form well suited to import into spreadsheet or graphics software. A number of parameters of interest can be exported, as well as the data for the performance maps (if available). Many of the figures in reference 2 were created using that capability.

The easiest way to get started with program AXCOM is to load one of the sample problems and experiment with the various menu options while being guided by the on-screen help. The program has a very broad range of options and capability, but all are rather easy to understand. The major input data sets used will be described briefly so the user can anticipate what is needed to create new input files. Note that gas thermodynamic equation of state data and input/output units are also required, as previously discussed.

Case Data Input Set: The first data set loaded is called the case data. This includes a title for use on output files, the number of stream surfaces from hub to shroud, the swirl specification type (C_θ or β), the mass flow or inlet volume flow, the rotation speed, the streamline curvature option (none, approximate or full, ref. 2, chapter 7) and the boundary layer blockage at the inlet station. An exhaust or exit loss coefficient and associated boundary layer blockage can be entered to correct the overall performance for the exhaust system loss (ref. 2, chapter 13). These two parameters are usually not independent, i.e., the exhaust loss coefficient usually depends on the blockage. Both are entered so that the program can compute the exit loss consistent with the way it was estimated. Then the total thermodynamic conditions and swirl parameter are entered at from one to five points across the annulus at the inlet station. Data are entered as a function of percent distance across the annulus (last entry must be at 100%). There is also an "Options" menu item in the case data window, where you can choose the options for boundary

layer blockage (neglect, specify or calculate) and for the blade performance Reynolds number corrections of chapter 6 (use or suppress). By default, the end-wall boundary layer analysis is used and Reynolds number corrections are suppressed.

Station Data Input Set: This data set defines the end-wall contours and the blade row type upstream of each station (none, rotor, stator or guide vane). If the annulus is being sized, only one contour can be defined (you will choose which one), along with the mean stream surface value of C_m and the angle between the local quasi-normal and the radial direction to be used at each station. The program may have both types of station data available if the annulus was already sized. But, if the annulus is re-sized, the coordinates of one wall will be changed. The type of data entered or edited in the station data input window is determined in the program's main window by checking or unchecking the check box for annulus sizing.

Blade Data Input Set: Blade row geometry data can be entered in any order. Consistency with the station data is not required. For example, you can enter data for all blade rows, even if not all station data has been loaded. Select the blade row type (rotor, stator or guide vane) and number (increasing sequentially from the inlet). Any stationary blade not following a rotor should be identified as a guide vane. Specify whether or not the blade row is shrouded (data required is different depending on this choice). Specify the number of blades and the location of maximum camber, a/c . For unshrouded blades supply the tip clearance. For shrouded blades specify the seal clearance, number of seal fins and the seal radius. Choose the option for specifying blade angles (θ and γ , κ_1 and θ , κ_1 and κ_2 , γ and C_{10} or κ_1 and C_{10}) and the blade profile and camberline types (ref. 2, chapter 4). Blade row geometry can be entered at up to ten radial positions. In addition to the angle data, above, you enter radius, chord and thickness-to-chord ratio. Data should be supplied to permit accurate interpolation over the required radius range. Before you can enter or edit data at radial points, you need to select the type of operation you want to accomplish. You can load all data, edit or delete existing data, copy data from another row already loaded, import it from a program AXSTAGE input file, rotate the blade (adjust γ), impose a radial shift on the blade data or scale all data. The input boxes for the geometry display any existing data for the selected blade row, but they are inactive until you select the operation you want. Many of these options are convenience features for standard-stage type compressors, to minimize the need to reload geometry already entered for another row. When blade data are loaded, the program defaults to the throat width empirical model discussed in ref. 2, chapter 4. The program can calculate the throat widths precisely for greater accuracy, which is recommended. When the throat widths are calculated, they are added to the input file for use on all future analyses. Don't forget to recalculate them if you change the blade data set.

Blockage Data Input Set: The usual practice is to compute the boundary layer blockage with the end-wall boundary layer analysis of chapter 8. But, you can enter blockage data directly, if you select that option in the case data window. Blockage data are entered at inlet and discharge stations and (optional) an interior station. Blockage at other stations will be computed by linear interpolation with respect to meridional distance. Data are the fraction of the annulus that is blocked by the boundary layers.

Stagger Angle Adjustment Data Input Set: You can enter data specifying the change in stagger angle to be imposed on the values loaded in the blade data set. The standard distributions of ref. 2, chapter 13 are available and offer the simplest method of entering variable inlet guide vane and stator data. Adjustments can also be imposed on any blade row as an absolute change in stagger angle. That is a good way to adjust rotor stagger angles when fine-tuning a standard stage design. Recall that stagger angles are normally negative for rotors and positive for stationary rows. You need to observe the same sign convention when entering your stagger angle adjustments.

16.0 Program AXSTAGE

Program AXSTAGE implements the axial-flow compressor stage design procedures described in ref. 2, chapter 10. This program works entirely with dimensionless data to design an ideal stage (rotor and stator) with the option to design inlet and exit guide vanes also. The stage designed is ideal in the sense that all blades are designed with the same hub and shroud radii and blade row losses are neglected. Hence, the flow field is isentropic and there is no rigorous mass balance accomplished and no equation of state is used. When the stage designed is applied in an

axial-flow compressor, the hub and shroud contours will have to be adjusted to maintain the same basic dimensionless flow field. The most common application for this program is the design of a standard stage for use in industrial axial-flow compressors as described in chapter 10. In that application, all stages will use identical blades, although the blade stagger angles may be adjusted to better match the specific applications.

Stage performance is specified in terms of rotor inlet mean flow coefficient, work coefficient, reaction and swirl vortex type. Some flexibility is available with regard to variation of the axial velocity through the stage and hub to shroud variation of the work input. The program will generate the ideal flow field based on the performance specifications and fit blades to the resulting velocity triangles that will operate at accepted design incidence angles. When the blade geometry has been generated and saved, it can be imported from this program's input file by program AXCOM to create the actual standard-stage compressor design by sizing the annulus to a desired meridional velocity distribution through the compressor and possibly adjusting stagger angles. This program will also supply detailed blade surface coordinates for drafting or manufacturing. When stacking blade sections to form a complete blade, the stacking axis is at the center of gravity of each section.

The program has the capability to operate in an automatic analysis mode, such that all calculations allowed by the available input data will be performed every time any input parameter is changed. That makes parametric studies, such as those discussed in chapter 10, rather fast and efficient. The program can cope with most errors that may be caused by bad data, so this rarely fails. But, the program can also run in manual mode if the user prefers to be more conservative. The program provides substantial tabular and graphical output that can be viewed on the monitor screen, inserted into an output file for the printer or exported in a text format appropriate to import into spreadsheet or graphics software. The export capability was used extensively to develop the figures in chapter 10.

The easiest way to become familiar with this program is to load the sample input file provided and experiment with the menus and design variables, using the on-screen help for guidance. It is a rather simple program to use and to understand. Many other sample problems can be easily set up from the cases discussed in chapter 10. There are basically six sets of input data required. A brief description of the input data sets is given here to help the user anticipate the data required.

Design Input Data Set: A title for use in output files must be entered. Select the blade profile and camberline types (chapter 4) for all blades to be included. Assign the hub, shroud and design radii. The flow field and blades are generated from hub to shroud using the performance specifications at the design radius from the other data sets.

Rotor Input Data Set: Specify the flow coefficient, work coefficient and reaction at the design radius and the swirl vortex type to be used. Also specify the axial velocity ratio across the blade row at the design radius and ratios of the work at the hub and the shroud relative to the design radius work (usually these three parameters will all be unity, but other values can be used if considered appropriate). The variable, K , of ref. 2, chapter 10 can be entered in place of work coefficient, if preferred.

Stator Input Data Set: Select the option to assign stator exit swirl and supply data needed for that option. Exit swirl can be assigned to produce a repeating stage, assigned at the design radius with a vortex exponent for other radii, or flow angles for the hub, design and shroud radii can be assigned. An axial velocity ratio across the blade can also be specified.

Inlet Guide Vane Input Data Set (Optional): Essentially identical to the stator data except swirl is specified at the blade inlet, and the repeating stage condition is replaced with a no-swirl condition.

Exit Guide Vane Input Data Set (Optional): Essentially identical to the stator data except that the repeating-stage condition is replaced with a no-swirl condition.

Blade Input Data Set: Supplies specific parameters needed to select the blades for each blade row being designed. The number of blades in each row is specified. Values of chord and thickness-to-chord ratio are supplied at the hub, design and shroud radii. For parabolic-arc camberlines, the location of maximum camber, a/c , is specified. For double-circular-arc blades, the ratio of the leading and trailing edge nose radii to the maximum thickness also must be specified.

17.0 Program AXDESIGN

Program AXDESIGN implements the multistage axial-flow compressor aerodynamic design procedures described in ref. 2, chapter 11. When the design has been generated, it can create a complete input file for use by program AXCOM for off-design performance analysis. It can also create input files for programs B2B2D and TDB2B for selected blade rows and stream surfaces and program FLOW3D for selected blade rows. That can be useful to obtain an internal flow analysis of blade sections or blade rows when you have concerns about their performance (e.g., if Mach numbers become super-critical, you might want to confirm that internal Mach numbers are not excessive).

Similar to program AXSTAGE, dimensionless stage performance data are used to define the desired flow field and blades are designed to match the resulting velocity triangles. However, it differs from program AXSTAGE in several important respects. It employs an equation of state with specified inlet thermodynamic conditions and rotation speed to convert dimensionless performance to a dimensional flow field. It uses conservation of mass to size the annulus to actually produce that flow field. It also includes the effect of blade row losses while calculating the flow field to provide an aerodynamic performance analysis of the compressor design at the design operating conditions. And, it accomplishes a complete multistage compressor design.

When sizing the annulus, stream surface curvature effects are ignored, as discussed in ref. 2, chapter 7. Also, a contour produced by annulus sizing is normally far from smooth enough to be acceptable. Program AXDESIGN allows the user to smooth this contour and repeat the design process with fixed end-wall contours. This will result in slight changes in the flow field, since conservation of mass now predicts the meridional velocity distribution through the compressor rather than sizing the annulus. But the blades will be designed to match the velocity triangles from this modified flow field. With fixed end-walls, the analysis can also include the approximate stream surface curvature model described in chapter 7. When carried out in this fashion, program AXDESIGN and AXCOM will yield almost exactly the same predicted flow field and compressor performance at the design

18.0 Program EXHAUST

Program EXHAUST is an aerodynamic performance analysis for the exhaust components of axial-flow compressors (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 (reference 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. Chapter 13 discusses the use of these performance parameters to define an exhaust loss coefficient to be used by program AXCOM to correct the predicted performance for exhaust losses. Flow conditions on the mean stream surface are also supplied through the diffuser passage, including a breakdown of the loss sources (friction, diffusion and curvature effects).

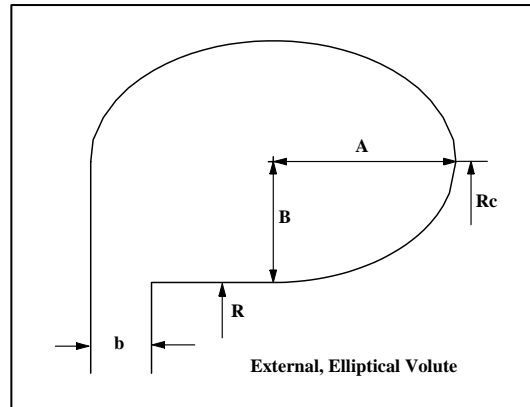
The input data for the exhaust diffuser includes the mass flow rate, inlet thermodynamic conditions and the fraction of the passage width occupied by the end-wall boundary layers at the inlet. The root-mean-square surface finish and the number of equal-spaced stations along the passage that are to be 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.

19.0 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 (entrance) boundary. This is typically obtained from a performance analysis by program CENCOM. 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 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 full-collection station (where the volute has collected 100% of the flow from the diffuser). But a passage area of zero cannot be manufactured, so a minimum area constraint is imposed. This is based on the entrance passage width, b , and requiring a reasonable curvature in the entrance 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.



20.0 Program BLADE

Program BLADE was developed specifically to validate the data and logic needed to model and display the standard airfoil types described in ref. 2, chapter 4 for their use in the various axial-flow compressor programs. It has been included in the *CompAero* system simply because it has often proven to be quite useful in supporting design and analysis activity. 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 airfoils and cascade form as applied in a compressor. 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. Many of the figures in reference 1 that display blade sections were created using this export function. This program is very simple to use and the on-screen help is quite sufficient to guide the user in all of its functions. It is expected that many users will find it very informative to actually view blades and cascades designed with the alternate camberline and profile styles available.

21.0 Program PTC10

Program PTC10 is a simple program to correct performance test results to the intended application conditions following the procedures of the ASME Power Test Code, PTC 10-1997 for centrifugal compressors. This includes cases where different working fluids are used for the test and the design conditions. It has been useful for consulting activity involving witness testing on behalf of the end user to provide an independent check on the performance corrections provided by the compressor supplier. The fluid mixture thermodynamic equation of state used is that of program RKM0D. Usually the supplier will use a different equation of state, so exact agreement with the supplier's calculations is not to be expected. But this is quite sufficient to identify any significant errors and to confirm the supplier's adherence to the principles of ASME PTC 10 in accomplishing a valid acceptance performance test. to confirm that performance guarantees have been satisfied.

22.0 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 *CompAero* data into Excel or in a “Save As” operation to provide Excel data for import into *CompAero*. 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 and 2, using the (*.dat) file type expected by PSI-Plot.

The *CompAero* 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, GASPETH, 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 12, 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) and a fairly generic geometry export for CAD/CAE/CFD applications. Users may want to check the online help for those export options to see if their content will be useful to support those types of applications.

Programs AXCOM and CENCOM 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 22) contain performance maps that were generated before this capability was added. Hence, they lack some of the data needed for this export.

23.0 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 changing it back just before printing would correct the problem (this can easily be done in the printed output window from the “Settings” menu). Presently, that step is now simulated 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. The user should make note of the manual fix in case it is needed. Normally the problem can be identified 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.

24.0 Sample Problems

When the software is installed, sample input files for the various programs are installed in sub-directory SAMPLES. The sample problems for the axial-flow compressor programs are specific cases included in reference 2. Centrifugal compressor programs sample input files are two hypothetical stage design problems referred to as Case #1 and Case #2. The base name of the input files identifies these cases, e.g., CASE1.CEN, CASE2.CEN, etc. Program RKM0D was not used in this process, so a representative sample input file SAMPLE.RK is included. The case definitions used for Program SIZE are as follows:

CASE #1: single-stage air compressor stage with axial inlet and "average" surface finishes.

Components: full-inducer, open impeller (no splitter blades), vaned diffuser and elliptical volute.
Working fluid: air modeled as pseudo-perfect gas (any equation of state is okay -- this just gives minimum computer running time and is adequate).
Inlet Conditions: total pressure = 100 kPa, total temperature = 300°K
Mass flow: set the inlet flow coefficient to 0.08.
Tip speed: rotational Mach number = 1.15.
Head: total-to-total pressure ratio = 2.5.
Impeller diameter: 30 cm.
Minimum (shaft) diameter: 7 cm.
Maximum (casing) diameter: unspecified.
Performance data style: adiabatic

CASE #2: low flow coefficient stage for multi-stage process compressors with polished surfaces.

Components: no-inducer, covered impeller (no splitter blades), vaneless diffuser and return channel.
Working fluid: propane modeled as pseudo-perfect gas formed from one of the non-ideal gas models (full non-ideal gas equation of state is also okay, but requires more computer running time).
Inlet Conditions: total pressure = 100 kPa, total temperature = 300°K
Mass flow: set the inlet flow coefficient to 0.02.
Tip speed: set the rotational Mach number to 0.4.
Head: use default head coefficient
Impeller diameter: 60 cm
Minimum (shaft) diameter: unspecified
Maximum (casing) diameter: unspecified.
Performance data style: polytropic

Most of these sample input files will be fairly self-explanatory, but a few comments may be of benefit. The blade-to-blade flow input files (CASE1.B2B and CASE2.B2B) are for the impeller shroud stream surfaces and were created directly by Program FLOW3D. The impeller curve files for Programs GASPETH or BEZIER (CASE1.BEZ and CASE2.BEZ) may be confusing unless the approach used is mentioned. The original set of curves (created by Program SIZE) is intact. The hub contour was copied before any modification. The shroud contour, and hub and shroud blade angle distributions (Case #1, only) were fit with Bezier polynomial before any modification. For Case #1, only the blade passage was modeled by these base curves (an option offered by Program SIZE). This is useful for full-inducer impellers to maintain the blade leading and trailing contour coordinates firmly fixed. Before attempting a Program FLOW3D, inlet and tip extensions are added to form curves denoted as "composite" curves. By contrast, Case #2 included the vaneless extensions in the base curves. The hub contour from program SIZE was copied before adjusting, and the shroud contour from Program SIZE was replaced with a circular-arc contour.

The reader may find it useful to try one or both of these designs from the above specifications and compare results with those from the sample problem input files. These designs were carried through to a reasonable level of quality, but both could be further refined. Each design was generated in about three hours, which is much less time than would have been used if the designs were to actually be manufactured and tested. About sixteen hours is a more typical centrifugal compressor stage design time with this design system. These cases are intended only to illustrate this

system.

References

1. Aungier, R. H., *Centrifugal Compressors: A Strategy for Aerodynamic Design and Analysis* (ASME Press, New York, 2000)
2. Aungier, R. H., *Axial-Flow Compressors: A Strategy for Aerodynamic Design and Analysis* (ASME Press, New York, 2003)
3. 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.
4. Aungier, R. H., *Turbine Aerodynamics: Axial-Flow and Radial-inflow Turbine Aerodynamic Design and Analysis* (ASME Press, New York, 2006)