POSTMARC

A Windows Postprocessor for CMARC and Pmarc

Version 7

1. System requirements and installation

Environment

POSTMARC runs under Windows, using the OpenGL graphics platform. Most graphics processors will handle OpenGL without difficulty, but if problems (such as incorrect hidden-surface and hidden-line elimination) are encountered with older processors it may be necessary to disable some graphics accelerations. To do this, go to *My Computer > Control Panel > Display > Settings > Advanced > Troubleshoot* and move the slider left to the second detent.

To suppress the legal agreement dialog that appears at the start of a program run, set the environment variable PSW_AGREE_YES by going to the Windows Control Panel, the *System* icon, the *Advanced* tab, and the *Environment Variables* button.

Installation

Normally, installation is handled automatically by the distributed installation program and the principal executable files are automatically added to the Windows Start menu. POSTMARC should be installed in the same folder as CMARC and/or DWT. The folder should also contain a copy of CMARC.DIM.

Once POSTMARC has been installed in the desired folder, you can place an icon on the desktop by locating the POSTMARC executable (POSTMARC.EXE) with the Windows Explorer, right-clicking to create a shortcut, and dragging the shortcut icon onto the desktop.

2. Data files

Input file types

POSTMARC reads both binary (.BIN) and formatted ASCII (.FMT) plot files produced by Pmarc or CMARC. Both types of file contain the same information, but binary files take up less disk space and load more quickly. On the other hand, ASCII files are more easily portable between operating systems, and it is simpler to write programs to recover information from them. So far as information mining is concerned, however, most of the information in the .BIN/.FMT file can also be retrieved from the ASCII .OUT file.

Which type of output file CMARC produces is controlled by the LPLTYP flag in the input file. The flag may be overridden from the command line in the DOS version of CMARC; otherwise, it must be edited in the .IN file. For binary output, which is the default, LPLTYP should be zero.

POSTMARC automatically identifies the type of the input file, the .BIN file having precedence in cases where both a .BIN and a .FMT file have the same name. All .BIN and .FMT files in the default directory are displayed in the file selection window.

POSTMARC also uses or creates other files, with extensions such as .PM, .ONB, .OFB, .RVS, .CVS, .PMX, and so on. The .PM file, which is generated by CMARC, is the basic source of data used for post-computing streamlines, boundary-layer characteristics, velocity scans, and so on. None of these files is created by Pmarc.

None of POSTMARC's input files, except CMARC.DIM, is intended to be modified by the user.

3. View management

Types of depiction

The model and its wake(s) may be displayed as a transparent wireframe or as opaque 3D surfaces with hidden lines and surfaces eliminated. To select the opaque 3D display, which is the most generally useful one, select *View > Hidden Line*.

To return to either of the basic depiction types after mapping colors or other information onto the model surface, hit `<Esc>`.

Setting the model's position

A model initially appears in top view. Several methods are available for viewing the model from different angles. They are 1) rotating the model in roll, pitch, and yaw, with the function keys or with the mouse; 2) directly setting an orthogonal view or a default perspective view; and 3) setting the direction cosines of an eye position with respect to the model coordinate system.

The model is not automatically kept in the center of the screen when it is rotated. To move it, hold down the right mouse button and drag the model.

Rotating the model with function keys

The model can be rotated with the function keys `F5` through `F10`, which control rotation about the X, Y, and Z axes of the model (`F5`, `F6`, `F7`) and the screen (`F8`, `F9`, `F10`) respectively. Hidden lines may be removed to resolve ambiguity. The `<Shift>` key reverses the direction of rotation. Positive (unshifted) rotation follows the right-hand rule; that is, when the right hand is held with the thumb pointing in a positive direction along an axis, rotation about that axis is positive in the direction the fingers curl.

To return the model to its initial position, click on the "reset" button.

Rotating the model with the mouse

Two sets of conventions are provided for rotating the model with the mouse. One set, selectable by *View > LMP View Logic*, is similar to those used by LOFTSMAN; the other is similar to those used by the original version of POSTMARC, and requires holding F3 (while moving the mouse) to perform rotations, F2 to zoom, and F1 to move the model.

LMP View Logic

The LOFTSMAN viewing system is based on the idea of a transparent globe with the model at its center. The observer is sprawled on the surface of this globe, head north, peering inward. Mouse movement controls the latitude and longitude of the observer.

In this scheme, the model is always upright; that is, its Z axis passes though the north pole of the globe. To rotate the entire view, move the mouse on the left-right axis while depressing both buttons; this has the effect of rotating the observer around his own eye(s).

Orthogonal views

As a shortcut to the three standard orthogonal views, the X, Y, and Z keys can be used. The lowercase letters produce views looking in a negative direction, or toward the origin, along the corresponding axes (*ie* rear, starboard, and top views); uppercase produce views looking in a positive direction or away from the origin (*ie* front, port, and bottom views).

Setting a viewpoint

The button with the eye icon is the "viewpoint" control. It is used to set the view direction. The dialog box requests three values. These are not angles, but rather *x,y,z* coordinates of the point in space, with respect to the global origin, at which the viewer's eye is located. For example, eye location [1 0 0] will be looking forward (toward the origin) along the X axis. Location [-1 0 0] looks aft along the X

axis toward the origin. Positive Z values indicate an eye position above the plane of the model. Positive Y values indicate a position to the model's right or starboard side. Coordinates may be fractional, but their absolute size is not important; rather, their relative sizes determine the angle from which the model is viewed. Thus, [10 5 4] produces the same view as [1.0 0.5 0.4]. Perspective convergence has not been implemented, and so distance from the origin is irrelevant.

While viewpoint is not a very intuitive method of positioning an object, it does have the advantage of providing a numerical equivalent of a view so that it can be duplicated at a later time or in different programs that provide this method.

Default perspective viewpoint

To obtain a default perspective view, hit `p` or `Shift-p`.

Stored views

In addition to the three orthogonal views and the default perspective view, POSTMARC will store up to 20 "key views" which are accessible using the digit and shifted-digit keys.

To store a view, position the model, then select

View > Key Views

and click the *V* button next to an unused digit. Henceforth (or until you store another view in the same location) typing that digit will produce that view.

Magnification

The button whose icon resembles a magnifying glass controls magnification. Normally the default magnification (1.0) scales the model to fit within the screen. A factor larger than 1.0 enlarges the model and one smaller than 1.0 shrinks it. Negative numbers are meaningless and are not accepted.

Zooming

To zoom, first click the zoom button, then move the mouse to one corner of the desired viewing area and, with the left mouse button depressed, drag a box around the part of the model at which a closer look is desired.

To unzoom, click on the "reset" button.

Use the magnification button, rather than the reset button, to partially reduce the magnification of a zoomed view.

An alternative method of controlling magnification and zooming is provided by the <F2> key when LMP View Logic is not being used. Vertical motion of the mouse with the <F2> key held down makes the model get larger and smaller.

The scroll wheel on the mouse may also be used to control zooming.

Panning

To move the model to a different location, drag it while pressing the right mouse button.

Solid colors

When hidden-line removal is used, model surfaces may be colored. Select *View > Colors* from the menu.

You can set different colors for the outer and inner surfaces of the model -- indispensable for checking patch orientation -- and for the "tops" and "bottoms" of wakes. On the model, positive corresponds to the outer surface. On wakes, the distinction between top and bottom is meaningless, but surface types should be consistent throughout any given wake sheet.

Patch selection

Any patch or group of patches may be selected or deselected for display. Click on *View > Patch Selection.*

To look at the inside of a duct, for example, the outer portion of the model can be
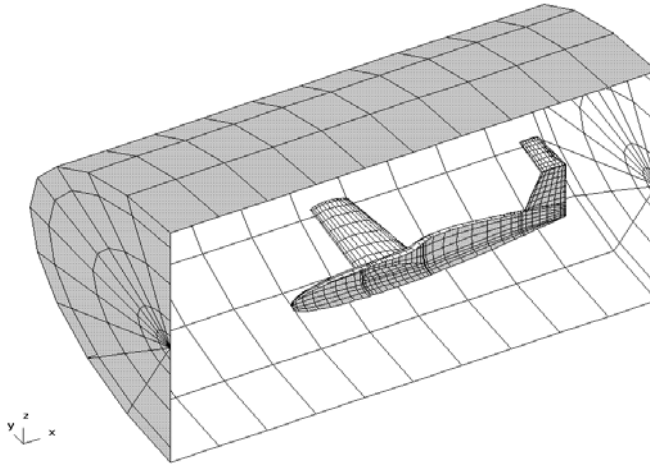
removed, provided that it has been defined as a separate patch. Patch selection is also important in obtaining integrated force and moment data for portions of a model. Only the portions of the model that are visible on the screen are included in the summation.

Wakes

To display one or all wakes, select

Display > Display wake(s)
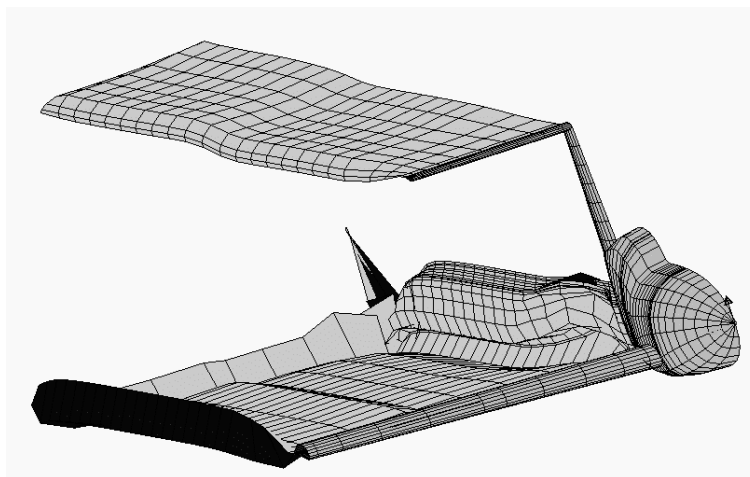
Like the model surface, wakes are displayed as a wireframe or with hidden lines removed, depending on the selection in the *View* menu.

To animate the wake(s), select

Display > Animate wake

or click on the wake icon in the toolbar. Wakes are numbered in the order in which they appear in the input file. Highlight the wakes you want to see. Wake animation continues to cycle until you hit <Esc>.

Saving wakes

It some cases, such as analyses of multi-element airfoils, it is quicker and simpler to have CMARC generate the wakes (INITIAL=0) than for the user to define them. This is still a long process, however, requiring about 20 time steps, and it would have to be repeated for each run of CMARC.

Once a wake has been defined, POSTMARC can file it in the proper format for inclusion in a CMARC input deck. When a developed wake has been edited into an .IN file, additional cases can usually be run without any time stepping at all so long as the angle of attack is not too far different from the original one.

Select

Calculations > Write Wakes

Normally, the wake shape at the final time step should be used, but only after checking that it is well formed.

By checking *Write separation line only* at the bottom of the form, you can make POSTMARC write into a .WSL file the node coordinates for a wake separation line. This in turn can be read by LOFTSMAN to produce a wake for geometries involving fuselage sides that are parallel or diverge behind the wing. For this purpose it is not necessary to fully develop a wake in CMARC; a single step is sufficient.

Displaying the wake separation line

When a CMARC input file includes a wake separation line and an analysis has been performed that includes wakes, the wake separation line can be displayed by checking *View > Wake Separation Line*. The WSL display alternates colors panel by panel to help you verify that it is not jumping panels.

Selecting and displaying panels with tilted vectors
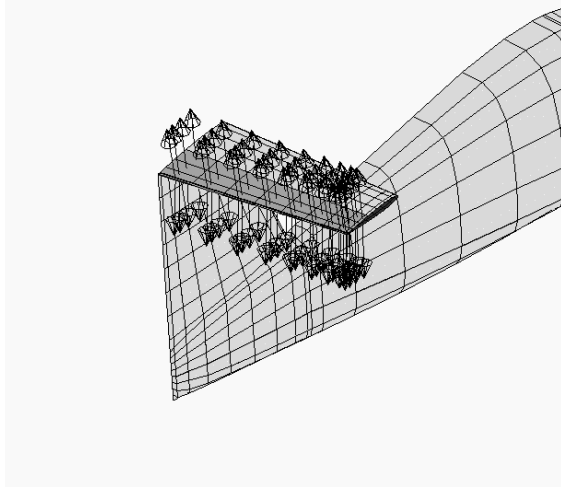
Tilting surface normal vectors is a convenient and accurate method of simulating control surface deflections up to angles of about 15 degrees. Panels with tilted vectors are identified on the &BINP11A line of the CMARC input file; see the CMARC documentation for a fuller discussion of this procedure.

Panels with tilted normal vectors can be both selected and displayed in POSTMARC.

To display panels, select *View > Tilt Panels > Manage/Write*. Check *View Tilt Panels* and supply the requested information for whichever source you have chosen.

Three sources are possible. First, you can define a panel or group of panels by entering data in the area captioned *Manual Input*. The items here are the same as those in line &BINP11A of the CMARC input file; groups of panels are defined by their first and last rows and columns. Check *View Tilt Panels* and *Manual Input* to highlight selected panels. Highlighted panels are visible only in the hidden line mode.

Second, you can display all or some of the panels defined in the CMARC input file. Click on *Load .in*. If the name of the appropriate input file is different from that of the file currently loaded, you must enter it before selecting *Load .in*. The list box is populated with all of the tilted panels defined in the input file. You can select from among them (or select all of them), and then click on *View from Cmarc/DWT* .

Finally, you can manually select panels and have POSTMARC write your &BINP11A line for you. Select *View > Display Tilt Panels > Pick panels* and select panels by placing the crosshair over them and doubleclicking the left mouse button. Hit <Enter> to finish the process, or <Esc> to abort it. Then return to the *Display/Create Tilt Panels* dialog, select *Cursor picked* and *Create CMARC cards*, and provide a file name. This file will contain a &BINP11A line ready to edit into the CMARC input file.

4. Displays of pressure and velocity

Selection of units

The basic inviscid analysis is independent of Reynolds number. Boundary-layer analyses, on the other hand, require that you specify fluid density and viscosity. To set these values, select

File > Units

Then select the appropriate fluid and, in the case of air, enter the altitude. The density and viscosity are displayed and become the default values.

Contour plots

The following potential-flow properties can be mapped in colors on the model:

Pressure coefficient
Gauge pressure
Velocity magnitude
Mach number

In addition, after a boundary layer analysis has been performed by "coating" the model with streamlines, the following properties may also be mapped:

Boundary layer thickness
Displacement thickness
Shape factor
Skin friction coefficient
Laminar transition

In addition, a two-color plot is available to identify areas where BL data have not been found. These may be areas of separated flow or areas where streamlines failed to be calculated for some technical reason such as a too-sharp bend between two panels.

Thicknesses are given in geometry units. Pressure and velocity coefficients are easy to understand, but in order for Mach number to have comprehensible dimensions, the values of VSOUND and VINF in the CMARC input file should

reflect their proper ratio. Thus, if the dimensions of a model are in inches and the flight speed of the model for a given analysis is actually going to be 4,000 inches per second, but VINF is set at 1.0, then VSOUND should be set at 13,392/4000, or 3.348. In other words, VSOUND should be set to the speed of sound in geometry units multiplied by VINF and then divided by the actual flight speed in geometry units.

To set contouring parameters, click on the *Display Options* button or select *Display > Options* from the menu.

The dialog presents a number of options:

Fringe contours: Raw data from CMARC provide a single pressure for each panel. When fringe contours are turned on, pressures are interpolated across panels, producing smooth transitions that more closely approximate the actual variations of pressure distribution. Fringe contours are on by default.

Include Panel boundaries: Panel boundaries may be indicated or omitted. Since they help to convey the shape of the model, they are on by default.

Automatic levels: The range of values represented by the color spectrum may be adjusted by the user, or may be selected by POSTMARC.
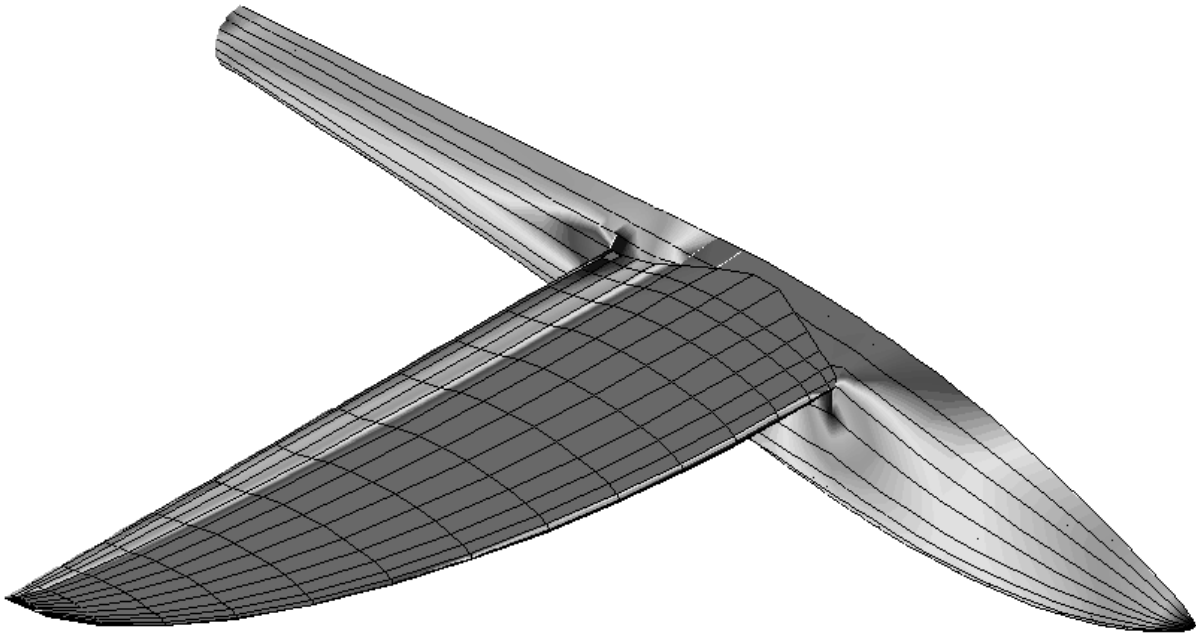
The advantage of having the program set the range is that you can switch among different types of output, such as pressures and velocities, whose values are of different magnitudes, without having to manually readjust the color range after each change.

The disadvantage is that many analyses that are otherwise valid include a few unrealistic values which may distort the range, compressing the useful band in an undesirable way. To protect against these outliers, POSTMARC can omit some values at the extremes. The size of the clipped area, as a percentage of the total number of values, can be set by the user (*% Clipping*).

When levels are set automatically, the limits can be rounded to convenient values. The effect is to enlarge the displayed range somewhat, without going to the extremes that outliers may cause. Check *Smooth Min/Max*.

Levels may be set manually to emphasize particular information. For example, a very narrow range of Cp values, such as -0.01 to +0.01, could be used to locate areas on a fuselage where pressure is close to static. The realistic range of Cp values for aircraft is from -5.0 to +1.0, but a narrower range, for example from -0.4 to +0.4, is usually more informative.

Symmetry: The model may be mirrored about the zero plane in any axis. For models that are symmetrical about the vertical plane, as aircraft normally are, the plane of reflection is the XZ plane. The default is None.

Number of colors: Larger numbers provide a smoother appearance. The largest permissible entry is 255.

% line offset: In order to ensure that lines representing contours, onbody streamlines, or surface force vectors are visible, they are drawn slightly above the model surface. If lines appear dashed, increase the value of the offset. The default is 0.01. This value also applies to highlighting, for instance when you click on panels to select them.

When you have made all your selections, click on the plot icon (rainbow oval) or select *Display > Contour Solid* to display the spectrum plot.

To return to the uncontoured hidden-line depiction of the model, hit <Esc>.

Contour lines

Pressure, velocity and Mach maps can also be displayed as contour lines rather than filled and blended colors. Click the *Lines Contour* button or select *Display > Contour Lines*. Relatively few colors should be used in order to avoid proliferation of lines of practically the same color and excessive crowding of lines in areas of rapid pressure change.

Screen XY Plots

After a contour map has been drawn, an XY plot of the variation in pressure, velocity or Mach number along any cross section may be obtained. The vertical range of the graph will be the same as the one selected for the spectrum plot unless you instruct POSTMARC to scale the plot automatically.

To define a section, display any view of the model and click on the cross-section button or select *Display > Screen XY Plot*. Then move the crosshair cursor to the point at which the section is to begin. Press the left mouse button, drag a line to the other end of the desired section, and release the mouse button. Coordinates of the cursor are continuously displayed at the bottom of the work area.

"Top" and/or "bottom" surface contours may be selected; these are in fact the surfaces nearer to and farther from the viewer. If only one line appears, the reason is most likely that both contours are identical, and one has overwritten the other.

Another option is a line representing the sum or difference of the "top" and the "bottom." When the surface in question is a wing, this line gives an approximate representation of the chordwise lift distribution. It should be kept in mind, however, that pressures act normal to the surface whereas lift is normal to the wind axis; and so the Y values represented by the "sum" line are not exact unless both top and bottom surfaces at a given station are horizontal or early so.
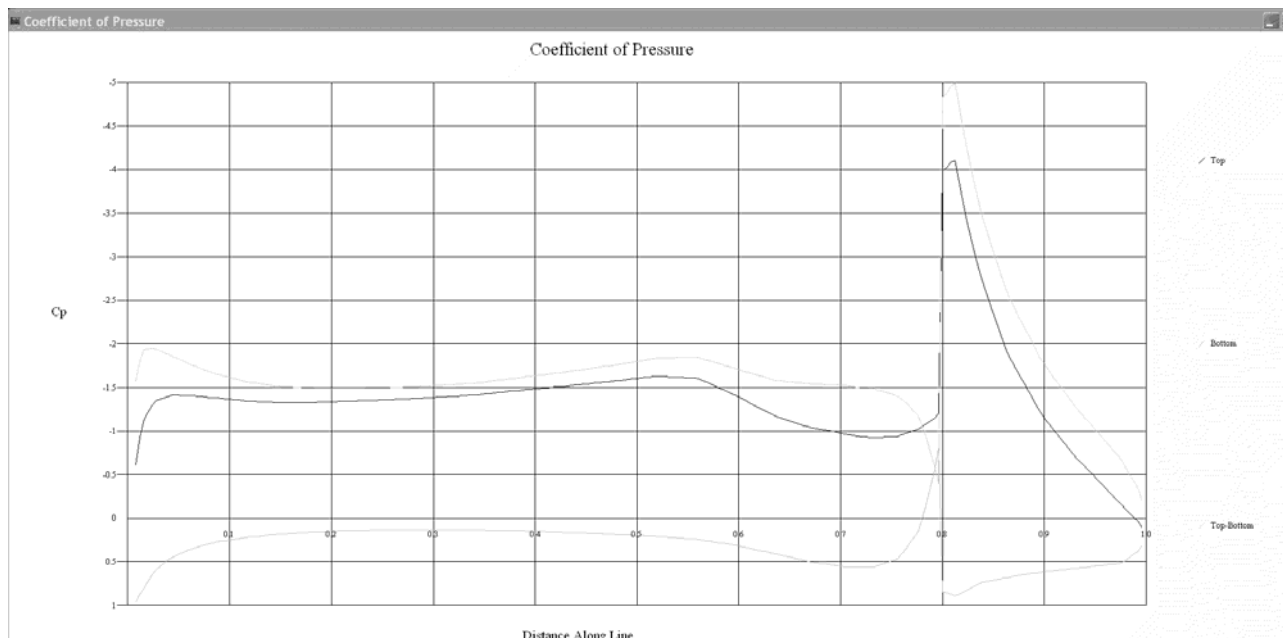
The raw data on which XY plots are based may be saved in tabular form. Check *Save to XY*.

POSTMARC provides the option of saving and displaying XY data in Excel. If you want to use Excel for this purpose, check that option in the *View* submenu. You may optionally open a new or an existing Excel file or close a file currently in use. If you do not select any of these items, POSTMARC automatically opens a new instance of Excel for XY plotting.

Differencing

POSTMARC will map the differences between two models in any plottable parameter. The models must be similarly paneled, but can differ in dimensions and/or in flight conditions. They can even differ in shape, so long as the patch count, and the number of rows and columns within each patch, are the same.

To load a second file for differencing, select

<div align="center">

File > Open secondary

</div>

After you make a selection, all commands related to pressure, velocity, and Mach number (including pressure distribution graphs and integrated forces and moments) relate to the two files together.

The default operation is second file minus first file. You can select first-minus-second or even first-plus-second by clicking on *File > Two File Operator*.

To turn off differencing, select *File > Two File Operator* and check *None*.

Mapping Cp to imported meshes

Loads may be mapped to either nodes or element centroids of non-CMARC FEA meshes. In either case, the points for which loads are desired must be supplied in the form of a simple ASCII list of coordinate triplets.

Select *Derived Data > Map Results to FEM*. Select a time state and a tolerance; POSTMARC will flag, for your information, all points for which the given point differs from the corresponding location on the surface of CMARC model surface by more than the tolerance. Provide the name of the input file containing the FEM point list and click on *Map Cp to FEM*.

POSTMARC produces a file in which the input coordinate list is duplicated, with the assigned pressure added alongside each point's coordinates.

5. Surface flow direction and streamlines

Local velocity vectors

Direction and velocity of local surface flow may be displayed as arrows originating from the centroid of each panel.
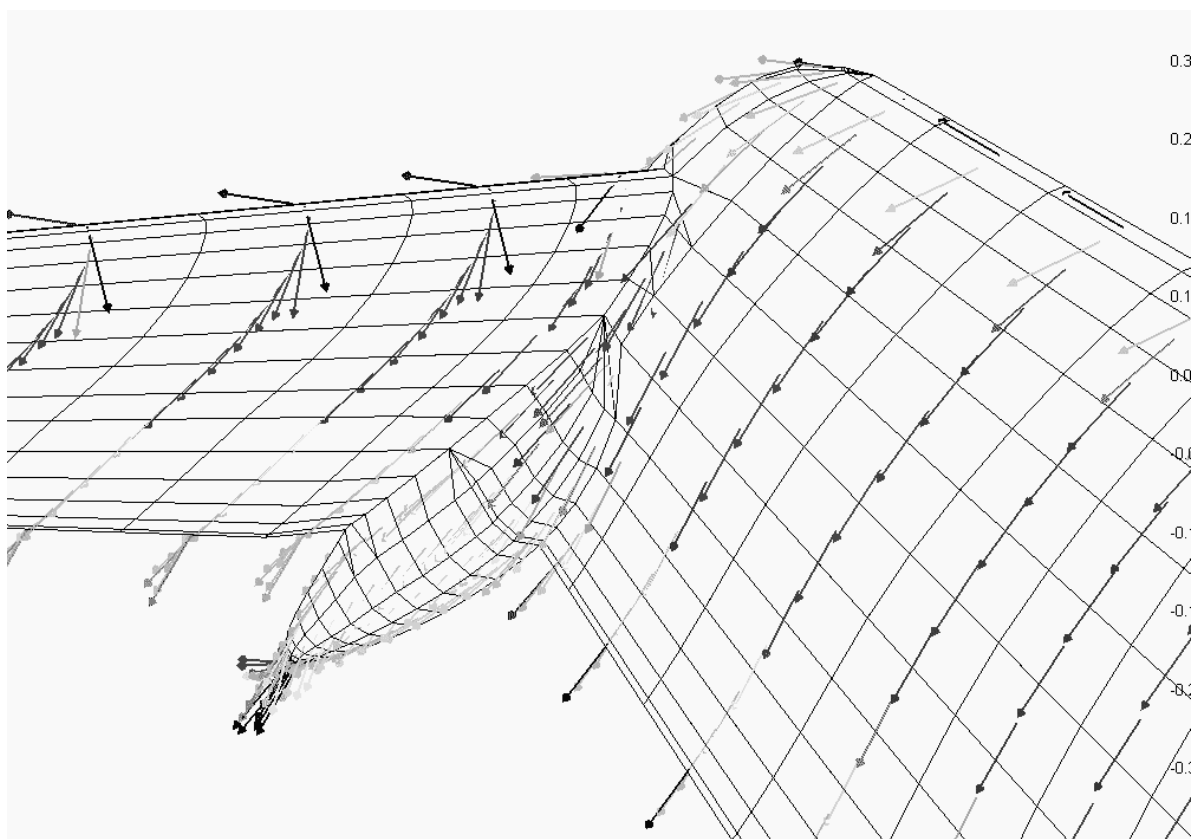
The direction of the arrow indicates the orientation of the local flow. The color of the arrow indicates the flow speed, pressure or Mach number. Arrow length indicates nothing, but can be adjusted in the *Display > Options* dialog under *Vector Display*.

To enhance the visibility of arrows, the model surface is made white.

Displaying on-body streamlines

On-body streamlines originate from the centroids of specified panels and are computed upstream or and downstream until the end of the body or a stagnation point is reached or turbulent separation is predicted. In real life, separated turbulent flow may in some cases reattach farther downstream, but turbulent reattachment will not occur in the numerical simulation.
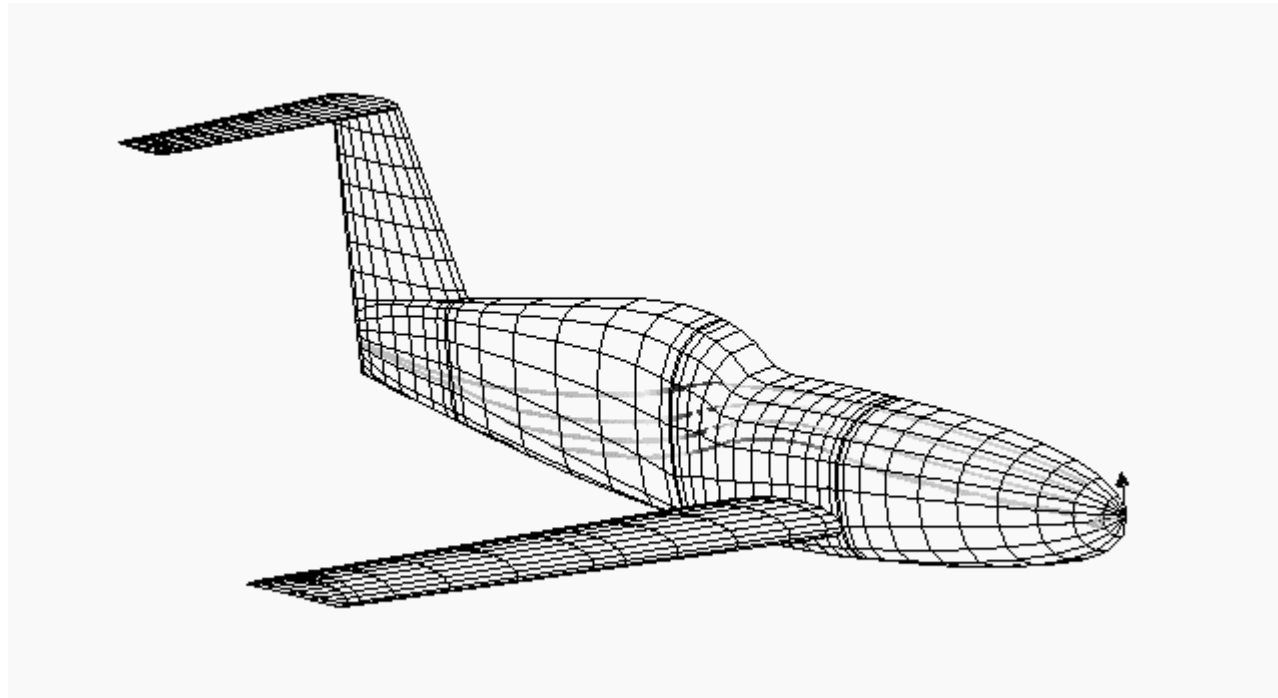
The colors of streamlines may encode a number of boundary layer conditions in addition to the usual pressure and velocity. These are:

    B.L. thickness
    Displacement thickness
    Shape factor
    Local skin friction coefficient
    Streamline state and thickness

When *Streamline state* is selected, the predicted point of laminar transition is indicated by a black dot.

    1) Select a display option
    2) Choose a range of streamlines *eg* 1,5
    3) Choose ALL streamlines *eg* *

For more prominent display of streamlines on printed output or on high-resolution displays, the width of the streamline image in pixels can be adjusted.

Generating streamlines in POSTMARC

The CMARC input file provides for definition of on-body streamlines with specified flow conditions (Reynolds number and kinematic viscosity). It is inconvenient to preselect streamlines, however, both because it is cumbersome to obtain ahead of time the ID numbers of panels through which the desired streamlines will pass, and because the analysis must be repeated for each new set of boundary-layer conditions.

POSTMARC, however, can compute streamlines for any or all panels, and for any boundary layer parameters, after a single CMARC run. The information used by POSTMARC is stored by CMARC in a file with the extension .PM.

A binary .PM file is automatically created by CMARC. CMARC can be made to write an ASCII .PM file instead, for instance if portability between UNIX and Windows systems is desired, by checking *Write Postmarc data in ASCII format* or by adding -pa to the DOS command line. POSTMARC automatically determines the type of .PM files when it reads them.

Selecting streamline locations

From the menu select

Calculations > Create On Body > Pick Panel Passing

Place the cursor over each panel through which you want a streamline to pass and double-click the left mouse button. The panel is highlighted. When you have finished, hit <Enter>.

Since onbody streamlines are calculated both forward and backward from the point you select, a single panel defines a streamline that may run the entire length of the model. If you wish to cover the entire body with streamlines, do not pick panels; go directly to *Manage/Calculate*.

Calculating on-body streamlines

When one or more streamlines have been identified, select

Calculations > Create On Body > Manage/calculate

A dialog box appears. If you have picked panels manually, they are listed; some or all of these may be chosen for processing. If you want to ensure that at least one streamline passes through every panel on the model, do not select any cursor-picked panels, but check *Cross All Panels* instead.

Processing of boundary layer data is optional. If you want a full boundary layer analysis, you must check *Include Boundary Layer Calculation.* If you do not supply values for Reynolds number and kinematic viscosity, POSTMARC looks in the .PM file for the values, if any, that were specified in the CMARC input file. If there are no values in the input file, or NBLIT is not set to 1, POSTMARC will warn that the values it found in the .PM file are unreasonable. In this case, enter new values in the dialog box before proceeding.

Calculated streamlines are appended to a file with the same name as the .PM file (which is the same as the name of the input file) but having an .ONB extension. It is not possible to weed streamlines selectively out of this file. It is possible, however, to delete the file altogether with the button marked *Delete Streamlines.*

Forcing laminar transition

Laminar transition may be forced at a desired location. Select

View > BL Trip Geometry > Pick Points

Doubleclick at two or more locations to define the trip line. Hit `<Enter>` to terminate a line or `<Esc>` to abort. When you terminate a line, the *View/Create BL Transition Geometry* dialog appears. (This dialog may also be reached via *View > BL Trip Geometry > Manage/Write.*) It allows you to select line segments to display, and also to create BINP13 and BINP15 cards for editing into an input file. To make the trip line(s) visible, check *View BL Transition Geometry.*

To create separate trip lines in different locations, repeat the process.

Trips can be created for temporary use or made a permanent part of the input file. To incorporate a trip line in the input file, first define the lines, then click on *Create Cmarc Cards.* A text file will be created containing BINP13 and BINP15 lines ready to edit into the CMARC input file. Streamlines generated in CMARC will be affected by these trips unless you change the value of NBLTRIPSEGS to zero. Streamlines subsequently generated in POSTMARC, however, will be affected

only if you load and select the trip lines in the *View > BL Trip Geometry > Manage/Write* dialog.

Generating boundary layer data for the complete model

If you select *Cross All Panels*, POSTMARC will automatically generate a streamline to cross every panel in the model. Nearly all streamlines will cross more than one panel, and many panels will be crossed by more than one streamline. The resulting information density is equal to that in the basic pressure/velocity analysis.

Values derived from this operation can be integrated to obtain friction drag for the entire model or for selected patches, and to provide spectrum plots of any boundary layer parameter.

Displaying boundary layer characteristics

Boundary layer characteristics may be mapped over the entire model in the form of a spectrum plot by selecting the appropriate button in the *Solid Contour* dialog. Automatic levels may be used for all parameters.

Boundary layer separation criterion

In developing the boundary layer along a streamline, a numerical parameter is used to detect turbulent separation. The default value of this parameter is 0.02. CMARC, DWT and POSTMARC all allow changing this parameter.

The separation parameter is of limited generality. A value that results in a realistic prediction of separation on a wing does not give a realistic prediction for a body of revolution. It is not possible, therefore, for a single value to provide a satisfactory prediction of separation for a complete aircraft model. For a specific isolated component such as a wing or an airship hull, however, it is possible to control the onset and propagation of separation with the separation parameter. For a wing of typical planform and airfoil section, a value in the vicinity of 0.07 yields a plausible-looking separation pattern.

In order to select the separation criterion, it is necessary to know the expected separation behavior. When a value has been found that produces a reasonable-looking separation pattern, test conditions can be varied to investigate, for

example, the progression of the stall with increasing alpha, or the value can be applied to a nonlinear (viscous) analysis. The results can yield useful trend information, but should not be relied upon for absolute values of BL-influenced parameters such as friction drag.

XY plotting of on-body streamlines

To inspect the variation of any parameter along a streamline as a function of chordwise location, select *XY Plot* in the *Display > On Body Streamlines* dialog. Only one streamline can be XY-plotted at a time. Enter a streamline number in either the *from .bin file* (that is, calculated in CMARC) or the *from .onb file* (calculated in POSTMARC) field, depending on how the streamline was calculated in the first place.

The order in which streamlines are generated when coating the body is essentially random, and so it is not possible to know or guess the ordinal number of a desired streamline in a complete coverage. If you do not know the number of the streamline you want to XY plot, two courses are available. One possibility is to erase the .ONB file (in the *Calculations > Create On Body > Manage/calculate* dialog) or to temporarily rename it, and then generate a single streamline passing through the panel of interest and inspect it.

Alternatively, you can select a single streamline, generate it, and note the number of streamlines that existed previously, which is reported when the calculation ends. The new streamline will have been added to the end of the currently existing .ONB file, and its number will be one greater than the previous total.

The raw data on which XY plots are based may be saved in tabular form. Check *Save to XY*.

Preventing premature separation

A rapid rise in pressure, such as may occur at the base of a windshield or a spinner, will sometimes produce a "turbulent separation" signal. When such separation occurs in the simulation, the flow does not reattach downstream as it would in real life. If you see large areas of magenta extending to the tail in the Cf display (indicating zero Cf), this is the likely reason.

To prevent this problem, you can recontour the area where the spike occurs to smooth the transition from one surface slope to another. One or more off-body streamlines can be tested to suggest the shape the air "prefers." The effect of recontouring is not completely unrealistic; it simulates the effect of a pool of dead

or rotating air, which does, in fact, occur at the base of a windshield and in other similar situations.

If the nonlinear option is used to generate streamlines in CMARC/DWT, then streamlines can be made to "jump" over panels where a temporary separation occurs. See the CMARC/DWT manual for an explanation of panel jumps.

6. Off-body flow

Off-body streamlines from CMARC

The paths of arbitrarily selected off-body streamlines computed in CMARC can be displayed. Off-body streamlines originate from points in the space surrounding the model, and propagate upstream and downstream.

The coordinates of the point of origin and the limits and step size for propagation must have been included in the CMARC input file.

Off-body streamlines are color-coded to indicate local flow conditions. The parameters that can be coded are pressure coefficient, velocity magnitude, and Mach number.

Calculation of off-body streamlines by POSTMARC
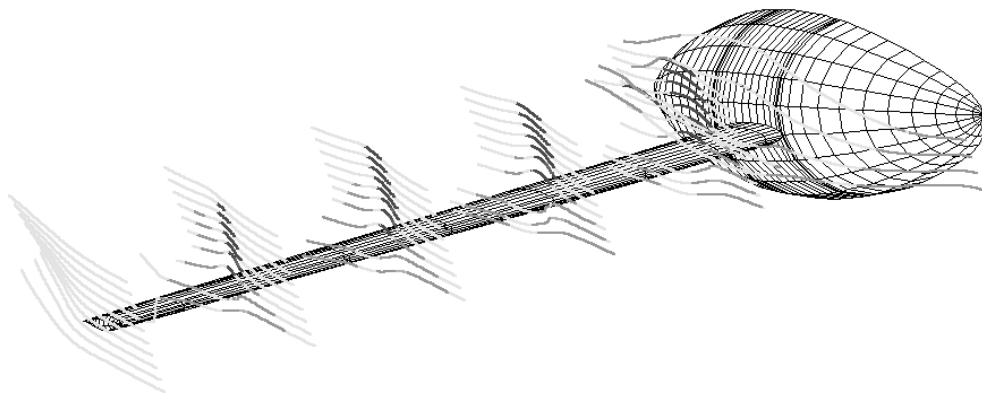
As in the CMARC input file, streamlines are defined by a point of origin, upstream and downstream distances, and the interval between points along the streamline.

Select

Calculations > Create Off-Body

A dialog box appears. The topmost items, up- and downstream distance (SU and SD), interval (DS), and the intersection-check toggle, are common to all off-body

streamlines. The intersection-checking toggle selects or deselects a routine that verifies at each station whether the streamline has penetrated the surface of the model. This operation somewhat slows down the calculation, and it can be omitted for streamlines sufficiently far from the body to be in no danger of colliding with it.

There are two basic choices: single and multiple streamlines.

Single streamlines

In the portion of the dialog headed *Single Streamlines*, enter three coordinates defining a point outside the model through which the streamline passes, and then click *Calculate Single Streamline*. When the streamline has been computed, click *OK*. The streamline may now be displayed by selecting *Display > Off Body Streamlines* or by pressing the *Off-body Streamlines* button on the toolbar.

Multiple streamlines

Multiple streamlines are defined by means of a rectangular grid of points.

The grid is defined by three points representing the corners of a parallelogram whose orientation in space is arbitrary. The first point defined, [X0 Y0 Z0], can be thought of as the origin of the parallelogram. The order of entry of the two remaining points is unconstrained, but most usually the second point, [X1 Y1 Z1], defines the horizontal extent of the parallelogram and the third point its vertical extent.

The parallelogram may collapse to a line when a set of coplanar streamlines is desired.

Below the second and third columns are fields labeled # *Points*. These represent the number of equally spaced points along the corresponding edge of the parallelogram. The total number of streamlines computed will be the product of these two numbers.

When the grid has been defined, click *Calculate Multiple Streamlines*. As with single streamlines, click *Done* when the calculation is complete.

Interactive selection of streamline locations

A convenient method of defining limited numbers of offbody streamlines is interactively, using the "wand" (so called because it mimics a wind tunnel smoke wand). This method requires first setting the X-axis station of a grid surface normal to the free-stream vector, and defining upstream and downstream distances and a step size, as in the other methods.

Select

Calculations > Create Off Body > Wand

Specify the wand plane location, the upstream and downstream lengths of the streamlines, and the interval size. Place the cursor on the grid and double-click the left mouse button. The streamline passing through that point is immediately calculated and displayed.

When you have finished calculating streamlines by the "wand" method, hit `<Enter>`. POSTMARC gives you an opportunity to save the streamlines.

You can rotate the model and perform various other operations while the "wand" is active.

XY plotting of off-body streamlines

To inspect the variation of Cp, velocity magnitude or Mach number along an off-body streamline, select *XY Plot* in the *Display > On Body Streamlines* dialog. Only one streamline can be XY-plotted at a time. Enter a streamline number in either the *from .bin file* (that is, calculated in CMARC) or the *from .ofb file* (calculated in POSTMARC) field, depending on how the streamline was calculated in the first place.

Streamlines are numbered in the order in which they were defined. If you do not know the number of the streamline you want to XY plot, two courses are available. One possibility is to erase the .OFB file (in the *Calculations > Create Off Body > Manage/calculate* dialog) or to temporarily rename it, and then generate a single streamline passing through the panel of interest and inspect it.

Alternatively, you can define a single streamline, generate it, and note the number of streamlines that existed previously, which is reported when the calculation ends. The new streamline will have been added to the end of the currently existing .OFB file, and its number will be one greater than the previous total.

The raw data on which XY plots are based may be saved in tabular form. Check *Save to XY*.

Rectangular and cylindrical velocity scans

Flow speed, direction and pressure may be displayed at three-dimensional grids of off-body points defined either in the CMARC input file or in POSTMARC. Grids may take the general form of a cylinder or a parallel-sided hexahedron (that is, a sheared, or un-sheared, rectangular prism).

Velocity scans are defined by means of either a hexahedral volume throughout which points are arranged in parallel arrays, or a cylindrical volume in which point planes are arrayed radially. For the sake of clarity, and because this is the nomenclature used in Pmarc-12, hexahedral scan volumes will be referred to as "rectangular," even though their sides are not necessarily orthogonal.

To display scan volumes, click on the appropriate icon in the toolbar. The parameter being mapped to color and the color range are set, as usual, through the *Display Options* form (rainbow icon).

Scan volume data are always drawn from the last time step in the original CMARC run.

Scan volumes computed in POSTMARC

Velocity scans can be performed by POSTMARC using data saved in the .PM file.

As in CMARC, definition of a scan volume requires specifying the parameters of the "box" or cylinder to be scanned. In the case of the rectangular scan, these are four corners of the volume. In the cylindrical scan, they are the axial start and end of the cylindrical volume, its inner and outer radius, the number of radial rows of points and the number of points in each row.

Select

Calculations > Create Velocity Scans

Three further menu choices are offered: *Rectangular*, *Cylindrical*, and *File*.

Defining a rectangular scan volume

In the dialog box, enter the coordinates of one corner and three additional points representing the extents of the box in length, width and height. The sides of the box need not be parallel to the orthogonal planes.

The following example defines a scan volume around the wing of the POMARC sample file. The box is swept to contain the wing.

> *Origin*: X = 125, Y = 18, Z = 20
> *Point 1*: X = 230, Y = 18, Z = 20
> *Point 2*: X = 235, Y = 120, Z = 20
> *Point 3*: X = 125, Y = 18, Z = -20

In the fields captioned # *Points*, specify the number of points to be distributed evenly along each axis of the scan volume.

Check *Intersection calculation* to ensure that POSTMARC does not attempt to calculate values for points located inside the model. This is not necessary if the entire scan volume is clearly located outside the model.

After defining a scan volume, press *Calculate Scans*.

Saving scan volume data

If you check *Save to .dat*, the current scan volume data are written to a text file.

All scan volumes associated with a particular model are saved consecutively in the same file, which has a "DAT" extension. It is up to the user to remember which is which, or to find out by displaying them. The order of items for each point is the same as in the OUT file. It begins with three columns of array indices. Next come three columns of Cartesian coordinates, three columns of direction vector components, and, finally, three columns containing, respectively, velocity magnitude as a coefficient of VINF, pressure coefficient, and Mach number.
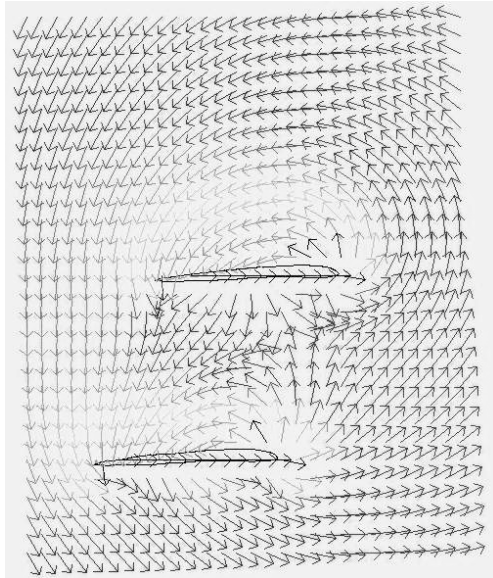
Single Point Calculation

Scan volume data can be obtained for single points by selecting

Calculations > Create Velocity Scans > Rectangular

Enter the coordinates of the point in the lower frame and click *Calculate Single Point.*

Displaying rectangular velocity scans

Velocity scans are displayed as arrows oriented with the local flow direction. The color of each arrow indicates the magnitude of one of the plottable parameters, namely Cp, Velocity magnitude, or Mach number. The length of the arrow has no meaning.

To contour the model surface while displaying a velocity scan, click *Contours - Solid.*

Optionally, the velocity of the free stream can be subtracted from each local velocity, so that the orientation of arrows represents the velocity increment created by the model. The image at left shows induced circulation around the wings of a Sopwith Camel.

Contouring scan planes

POSTMARC will create and display contour maps of pressure, velocity, or Mach number over one or more planes within a rectangular scan volume. Only planes containing points can be contoured.

After computing a scan volume, press the *Contour velocity scans (slices)* icon on the toolbar or select *Display > Contour Velocity Scans.*

Select the scan type and the parameter to be displayed. Select the scan to display by entering its sequential number in the DAT file. Click *Load # Pts* to bring the number of points in each scan plane into the form.

You may select one, several, or all planes along any axis. To contour the model in the same display, click *Contours - Solid* and select the parameter to be contoured.

Click *OK* to display the contours.
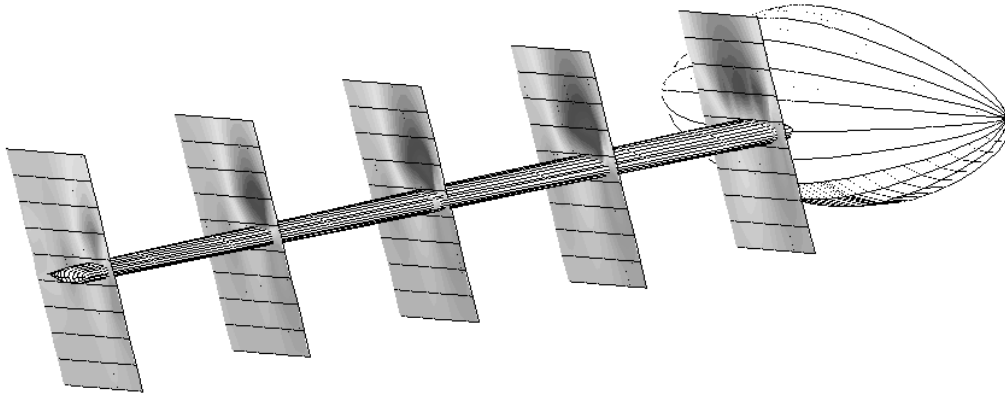
Cylindrical scan volumes defined in POSTMARC

The cylinder geometry is described as for CMARC input, and the data input form uses the same nomenclature.

Select

Calculations > Create Velocity Scans > Cylindrical

First, define the origin of the central axis ([X0 Y0 Z0]). This will normally be on the front surface of the cylinder. Next, define the other end of the central axis ([X1 Y1 Z1]).

Third, define a point ([X2 Y2 Z2]), on an arbitrary radius vector from the origin, that will serve as the zero line for angular measurements. Usually, this baseline vector will be vertical, pointing upward, and its X and Y coordinates will be the same as those of the origin. Its length is unimportant. For example, if the origin is at [130 37 -19] then the point defining the baseline vector might be [130 37 -18].

Next, define the inner and outer radii of the cylinder, R1 and R2. If the cylinder is solid, R1 will be zero.

Next, define the starting and ending angles with respect to the baseline vector. Angles are measured in degrees according to the right hand rule; that is, positive rotation is clockwise as viewed from the front. For a full cylinder the values would be zero and 360, but any wedge-shaped segment of the cylinder may be specified.

Finally, three values define the point density in the radial (NRAD), circumferential (NPHI), and axial (NLEN) directions. By setting either NLEN, NPHI, or NRAD to zero, the scan volume can be collapsed to a plane.

Intersection checking should be turned on if some portion of the model lies within the defined scan volume.

Cylindrical scan volume display

Like rectangular scan volumes, cylindrical ones can be displayed as groups of arrows showing stream orientation and color-coded for pressure, velocity, or Mach number, or as contoured planes.

Scans from imported point lists

Rather than define a volume geometrically, you can provide a filed list of points and POSTMARC will return scan information for each point. Select

Calculations > Create velocity scans > File

Check *Intersection calculation* if you think some of the points in the file may be inside the model.

Click on *Calculate scans* and select a file name. The input file should be a text file with 3 coordinates for each scan point, delimited by one or more spaces, on each line. There is no limit, other than available memory, to the number of points that may be processed or displayed.

POSTMARC creates a file with the name of the current input file and an .RVS extension, or appends to it if one already exists. Scans from filed points can be displayed in the same way as rectangular and cylindrical ones, except that they cannot be displayed as contoured planes.

7. Information about individual panels

Geometry and aerodynamic data (individual panels)

Place the cursor upon a panel and doubleclick the left mouse button. The patch, row, column, and corner node numbers are displayed in the status line across the bottom of the screen. If the display is contoured (*ie* it is a spectrum plot) then holding the Shift key while doubleclicking causes the values of Cp, velocity magnitude, Mach number and panel area to appear instead.

To obtain more detailed information, select

View > Detailed panel info

Now, when you doubleclick on a panel a form appears with all available information related to that panel.

Panel XY plots

Panel parameters can be plotted against time step. Since parameters are plotted against time, this procedure applies only to time-stepped analyses.

Select

Display > Time XY Plot > XY Plot on Panel

Double-click on the panels you want to examine. Hit <Enter> when you have finished.

The raw data on which XY plots are based may be saved in tabular form. Check *Save to XY*.

Integrated XY plots

A number of parameters, including lift coefficient, induced drag coefficient, span efficiency factor taken at the Trefftz plane, and force and moment coefficients about all three axes may be plotted against time step. Select

Time XY Plot > XY Plot (Other)

and select a parameter. Be sure to note the scale of the ordinate, which is set automatically to match the range of variation in the data. A quite jagged-looking plot of CL, for example, might convey the impression of a failure to converge satisfactorily; but the fluctuations could in fact be inconsequential, limited to the third digit to the right of the decimal point.

Another option is to plot spanwise distribution of the local lift coefficient (cl) along any lifting surface. (Postmarc identifies lifting surfaces as those for which IDPAT=1 in the input file.) Surfaces are identified in the scroll box by the labels they have in the input file.

A third option is to view the convergence history for a selected time step. This is the size of the residual error that is compared with SOLRES at the end of each time step to determine when to stop iterating. The plot is logarithmic and should appear as an initially steep, generally monotonic descent.

The raw data on which XY plots are based may be saved in tabular form. Check *Save to XY*.

8. Integrated forces and moments

Patch and air data selection

Forces and moments may be integrated either with respect to the aircraft coordinate system ("body axes") or with respect to the world ("wind axes").

The information given corresponds to the model as displayed. Thus, for example, to obtain a total normal force coefficient the entire model, not just half of it, must be displayed; alternatively, if half the model is displayed, the reference wing area (SREF) should also be halved. Forces and moments for isolated patches or for groups of patches may be obtained by selecting those patches and hiding others.

Click the *Sum Forces* button. Click the *Load .in* button to import certain reference values from the .IN file. If the .IN file's name is not the same as that of the .BIN file, you must enter the correct .IN file name before clicking *Load .in.*

Most values on the input side are provided for information, but some can be adjusted to change the values of forces and moments. The adjustable values are speed, density, area, semispan, wing chord, and the coordinates of the moment reference point.

The following nomenclature is used:

   Fx, Fy, Fz   Dimensional forces in the positive direction along the X, Y, and Z axes respectively. Units are consistent with those selected for fluid density.

   Mx, My, Mz   Dimensional moments about the respective axes. Mx is rolling moment, My pitching moment, Mz yawing moment.

   CN   Normal force coefficient with respect to body axes.

   CA   Longitudinal (drag) force coefficient with respect to body axes.

   CY   Side force coefficient with respect to body axes.

   C_m   Pitching moment coefficient.

   C_l   Rolling moment coefficient.

   C_n   Yawing moment coefficient.

Coefficients are referred to the wing area SREF, the reference chord CBAR, and the semispan SSPAN. Drag coefficient (CD), however, may optionally be obtained in terms of the wetted area by checking *CD wrt Swet* (*wrt* = "with respect to").

Fluid density must be expressed in units consistent with the model dimensions. The same is true of velocity (VINF), which must be entered in order to obtain forces in dimensional form. For example, if the model is dimensioned in inches, velocity must be expressed in inches per second, with a value of 3,500, for example, representing approximately 200 mph. The default velocity is 1.0 and the default density is that for a standard day at sea level.

Important

If the angle of attack or yaw angle supplied in the input file has been overridden in the CMARC analysis, the actual value(s) used must be manually entered here in order for the displayed results to be correct. Enter the correct values in the ALDEG and YAWDEG fields on the left side of the form.

Information for stress analysis

At the bottom of the *Sum Forces* window is a button labeled *Dimensional Load Diagrams.* It invokes a dialog dedicated to displaying loadings, shears, and bending and torsional moments needed for stress analysis. Note that these values represent the contributions of air loads only; they do not include contributions or relief from the mass of the model itself.

The initial step in obtaining load graphs is to define bending axes. There are two, and they share a common origin. The origin typically lies on the neutral axis of a principal structural member and at the root of the beam. The root may be an attachment point, such as a fitting connecting a wing to a fuselage, or it may be an arbitrarily selected station. Axes are defined as vectors by specifying a second point along the axis in terms of its offsets from the origin. Once they have been defined, both axes may be displayed in the depiction of the model if *View > Load Report Axes* is checked.

The first axis (*Bending about*) is the one about which the beam would rotate under load if it were not restrained. In the case of a wing beam, this would be a fore-aft line normal to the shear web.

The second axis (*Along*) is the one along which loads are plotted. In the case of a wing, this would be a spanwise line approximately along the beam's neutral axis.

Click on *Plot diagrams.*

POSTMARC displays four windows that may be separately maximized, minimized, or printed in graphic or tabular form.
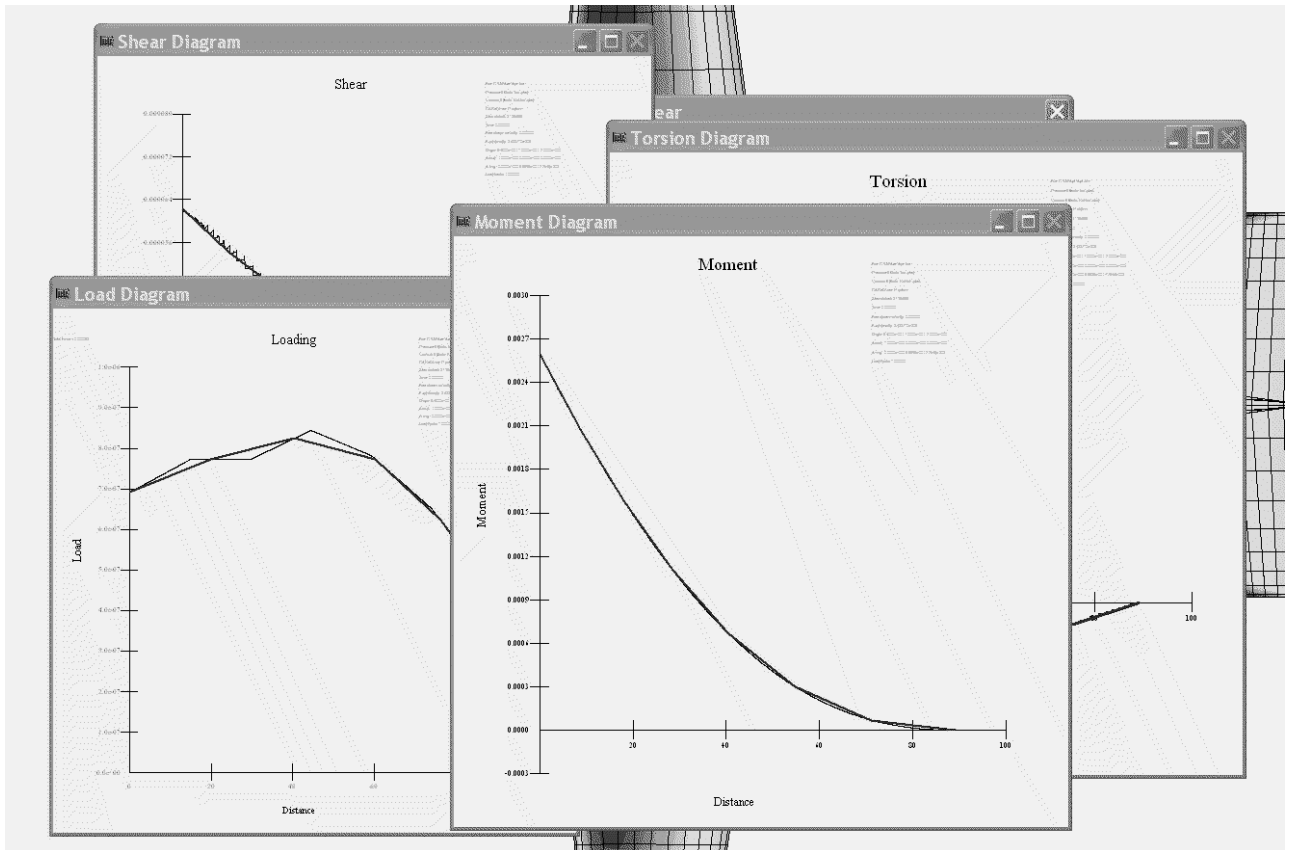
In order to integrate panel loadings, POSTMARC subdivides the surfaces in question into a regular pattern. In some cases the pattern of subdivision interferes with the pattern of the model's panels, and the loading diagram is jagged or irregular. The effect of these irregularities on shears and moments is small, but if you wish you can vary the subdivision parameters by varying the *Average % interval* (a real number). Processing time increases in inverse proportion to the *Average % interval.*

If *Continuous load line* is unchecked, the load graph is presented as raw data, that is, as a stepwise graph showing the average load level on each strip of width equal to the Average % interval. If *Continuous load line* is checked, the load graph appears as a continuously varying polyline instead.

You can elect to spline any of the graphs. In this case, smoothed values are represented by a heavy line.

To save all of the results at a specified interval in tabular form, enter an interval and click on *To File, every*.

Stability derivatives and lift curve slope using CMARC

Stability and other derivatives are most easily obtained using DIGITAL WIND TUNNEL, but a simplified version of the procedure can also be run manually.

To obtain stability derivatives for aircraft, it is necessary first of all to run cases at two angles of attack. For the sake of example longitudinal stability is used here, but the procedure for obtaining yaw and roll derivatives is analogous. (Note that to obtain yaw derivatives you must model the complete airframe, setting RSYM=1.0 on the basic input page and IPATCOP=1 for all patches. Wakes must be explicitly defined for mirrored panels; CMARC has no facility for mirror-copying wakes.)

A matrix of yaw and pitch cases can be run using CMARC's batch capability.

Since CMARC yields linear variations of moments with respect to angles of attack, it is not necessary to run more than two cases to obtain a result for any given parameter; but additional cases at different angles may be run to average out slight variations or to observe the effect of different yaw angles on longitudinal stability or vice versa.

To obtain the lift curve slope, obtain CLs at two angles of attack, preferably one degree apart.

The procedure for obtaining longitudinal stability derivatives is similar. Note the value of C_m for two different angles of attack. The difference between the two coefficients, divided by the difference in angles of attack, is the stability derivative for a CG position represented by the coordinates of the *Moments relative to* point.

To obtain yawing stability derivatives or rolling moments as a function of yaw, use a similar procedure with the values of C_n or C_l. Note that coefficients are reported with respect to both wind and body axes. Normally, the values with respect to wind axes should be used.

Maximum roll rate may be obtained with a fully defined bilaterally symmetrical model by setting the maximum deflection into the ailerons. Run two analyses, one with zero roll rate and one with, say, 100 deg/sec roll rate. The rolling moment coefficient diminishes as the roll rate increases, and the point at which the rolling moment is zero is the maximum roll rate.

Note that the roll rate is really a helix angle. If the roll rate is entered as its real value, eg 100 deg/sec, then the speed (VINF) must have its real value as well. If VINF is 1.0, then the roll rate must be divided by the speed to obtain a proportionally correct value.

An empirical correction (found in Hoerner, *Fluid Dynamic Lift*, and applied automatically by DWT) must be applied to convert the roll rate obtained into a realistic one. Rates obtained from POSTMARC are useful for comparing various configurations, but not for predicting actual roll rates.

Neutral point

To obtain the longitudinal static neutral point, use two cases with angles of attack differing by one degree. For each case, note C_m, then move the X coordinate of the moment reference point an appropriate distance aft and note C_m again. By subtracting the corresponding values and dividing by the distance between stations, obtain the slope of C_m with respect to alpha for two moment reference points. For longitudinally stable configurations, the more aft location of the moment reference point will correspond to the lesser of the two slopes.

Next, convert the slopes to angles by taking their arctangents.

Finally, obtain the neutral point by extrapolating from the angles to the station at which the angle would be zero. This may be done using interpolation utility (in the *Misc* menu) in LOFTSMAN.

Elevator angle to trim

A similar procedure may be used to determine elevator angle to trim at a given angle of attack and CG location. Use the CG location as the moment reference point and run analyses at two values of elevator tilt, preferably bracketing the expected one.

Stability analysis with DWT

All procedures related to static stability, trim, neutral point and roll rate are performed automatically, and with greater refinement, by DIGITAL WIND TUNNEL, AeroLogic's Windows executive for CMARC. DWT also predicts dynamic stability rates and convergence/divergence.

Inertial calculation

For calculation of dynamic stability with DIGITAL WIND TUNNEL, it is necessary to provide moments of inertia for the complete model. POSTMARC incorporates a procedure for doing this.

Select

Calculations > Inertial properties

A form for estimating inertial properties appears.

POSTMARC recognizes two categories of items contributing to moments of inertia: first, discrete, concentrated masses, and second, the surface of the model.

Inertia of discrete components

At the top of the *Calculate Inertial Properties* form is a box containing a scrolling master list of items contributing to the mass of the model. To add items to the list, enter their names, masses, and the coordinates of their centroids in the area captioned *Item Inertial Properties*. When you have defined an item, click on *Add item to list.* You can save the list at any time by clicking on *Write File*, and you can import an existing list with *Read File*. You can also delete individual items or all items in the list.

To compute the inertial properties of items on the list, you must provide a moment reference point. This point normally corresponds to a design CG location, and its coordinates are usually stored in the CMARC input file as RMPX, RMPY, and RMPZ. To retrieve these values from the current CMARC input file, click on *Load .in*. If the name of the appropriate input file is different from that of the current file, you must type it in the adjacent box before pressing *Load .in*.

The area captioned *Calculated Properties* delivers the aggregated mass, moment of inertia, and center of gravity of all components in the master list. Note that if the master list is fairly complete, it serves as a weight and CG buildup for the complete aircraft or vessel.

Note also that POSTMARC slightly underestimates total moments of inertia, because it does not take into account the rotational moments of inertia of individual components about their own centroids. For example, POSTMARC would find zero moments of inertia for an engine whose mass centroid coincides

with the moment reference point, but the engine would, in fact, contribute some damping to pitch, yaw and roll rates. If you wish to take these moments into account, you can enter moments manually in the *Item Inertial Properties* area.

Inertia of the model surface

The model surface is addressed in the area captioned *Patch Inertial Properties*. The approximate moment of inertia of the highlighted patches is obtained by entering an average value for the mass per unit of area of the surface and clicking on *Calculate Inertial Properties*. For example, the mass of .032 aluminum sheet is about .00323 pounds per square inch. By default, all surface patches are included; but you can obtain moments of inertia for portions of the surface by selecting or deselecting patches in the list box.

To add all or part of the surface to the master input list, provide a name and a *Weight/Area* ratio for it and click on *Add Item to List*. Be sure that the *Weight/Area* ratio is in units consistent with those used to define the model.

9. Output options

Hard copy

To obtain printed copies of screen displays, select *File > Print*. Use *File > Print setup* to gain access to the standard Windows printer setup forms.

POSTMARC will not print wakes that are being continuously generated by the wake animation routine.

Saving to the clipboard

Screen images can also be saved in the Windows Clipboard. Hit `<Alt-PrtSc>`. The current screen (including the toolbar) will be moved to the Clipboard.

POV-Ray

POV-Ray is short for Persistence of Vision Raytracer, a tool for producing high-quality computer graphics. POV-Ray is copyrighted freeware and can be downloaded from the World Wide Web at *www.povray.org*. Examples of POV-Ray rendered models can also be seen on the AeroLogic web page at *www.aerologic.com*.

To make a POV-Ray file of the current view, select

File > Export > POV-Ray

to create and name a file.

POSTMARC writes out every panel on a patch by patch basis. Each four-sided panel in the POSTMARC input model is broken into two triangles to conform to the "mesh of triangles" format of POV-Ray. POSTMARC assigns a different texture or color for each patch. The camera direction (or view position) is stored in the POV-Ray file. The POV-Ray deck can be manually edited in order to modify

textures and/or colors for each patch. Items such as landing gear, propellers, etc. can be added in order to make photorealistic pictures.

In general, what is saved in POV-Ray format is what you see on the screen in POSTMARC. If only half a model is displayed, only that half of the model will be saved.

VRML

VRML is an acronym for Virtual Reality Modeling Language. VRML allows you to describe 3D objects and combine them into scenes and "worlds." You can use VRML to create interactive simulations that incorporate animation, motion physics, and real-time multi-user participation.

To save a wireframe geometry, select

File > Export > VRML

POSTMARC will prompt you for a name and create a VRML file. As with POV-Ray files, only the portions of the model actually displayed on the screen will be saved. The default file extension is .WRL (for "world").

Models can also be saved as colored surfaces representing any of the contourable variables computed by CMARC or POSTMARC. Select

Display > Contour solid (VRML)

Evaluation copies of a VRML viewer, CosmoPlayer, can be downloaded from the web site at *www.vrml.sgi.com.* Examples of POSTMARC-generated VRML output can be seen on the AeroLogic web page at *www.aerologic.com.*

DXF

DXF is a graphics data storage format originated by the creators of AutoCAD and now supported by most CAD systems. POSTMARC will save the current model (as displayed on the screen) in DXF "mesh" format, which allows most CAD programs to perform hidden line removal and rendering.

Select

File > Export > DXF

The output file has a .DXF extension. As with other geometry-filing formats, only the portion of the model displayed on the screen will be saved.