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AEROSPACE DESIGNER'S TRAINING GUIDE
FOR PRO/ENGINEER

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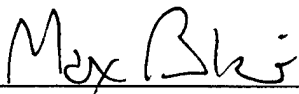AIR FORCE MATERIEL COMMAND
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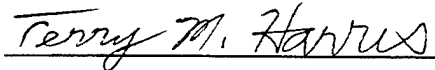
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THIS REPORT IS A TUTORIAL FOR THE GEOMETRIC DESIGNER OF AIRFRAMES AT THE CONCEPTUAL LEVEL WHO HAS A REQUIREMENT TO USE PRO/ENGINEER SOFTWARE. THE READER IS CAREFULLY GUIDED THROUGH THE STEPS TO DRAW A WING SURFACE, WING SOLID GEOMETRY AND FUSELAGE SURFACE GEOMETRY. THIS MATERIAL IS SUITABLE FOR ENGINEERS WHO HAVE ALREADY COMPLETED A BASIC TRAINING COURSE WITH PRO/ENGINEER. AFTER COMPLETING THE EXERCISES, THE READER MAY BE PREPARED TO WORK ALONE IN EXPLORING TECHNIQUES FOR DRAWING AEROSPACE GEOMETRY.

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FOREWORD

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The work was performed during the period of June 1995 through August 1995. Mr. Mark J. Cooper was the lead author.

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The author would also like to thank Ms. Vicki Lockwood, 1Lt. Chris Shearer, and Ms. Geetha Bharatram for testing the manual and providing feedback.

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Table of Contents

SECTION I

Introduction

This is a training guide for aerospace vehicle designers using Pro/ENGINEER Version 14.0. The guide is arranged in sections that take the user, step-by-step, through the creation of different features with Pro/ENGINEER. Pro/ENGINEER manuals should be read after each new feature is created. This will give the user a better general understanding of the method for creating that type of feature.

The first section is designed to get the user acquainted with the Pro/ENGINEER interface and some simple feature construction. After that, each section builds part of a conceptual design model of an aircraft.

It is best to follow the commands in the manual one at a time. If you read too far ahead, you may miss steps when you go back to actually carry them out. However, it is good to have an idea of the 'big picture' of what you are doing. Occasionally look forward to the next diagram, which will give a rough idea of what you are working towards. It could help clarify an otherwise confusing command. It may also be helpful to know what the next one or two steps are. Again, knowing what you are trying to accomplish may make the directions for getting there easier to understand.

Bold face text are actions to be taken by the reader. For instance, Pro/ENGINEER menu picks are denoted by **MENUTITLE(Menu_Choice)**. As you move through the sections, instructions for commands you have already carried out will be abbreviated. If you want detailed instructions, refer to a previous section or Pro/ENGINEER manuals.

In Pro/ENGINEER, sketches are used to define features. Features make up a part. Several parts can make an assembly. This manual will deal only with sketches, features, and parts. Keep this hierarchy in mind as you learn Pro/ENGINEER.

When you save files in Pro/ENGINEER, they will be saved to the directory that you were in when you started Pro/ENGINEER. You can change the directory from inside Pro/ENGINEER using **MODE(Misc) / MISC(Change_Dir)**. Before starting, you may want to create a directory for your Pro/ENGINEER files. Each time you save in Pro/ENGINEER, a separate file is created. Each file will have the same name but will be numbered as it was saved. This will allow you to recall a part or sketch you have been working on at any stage you saved. If you save often, many files will be created. This can take up a lot of memory. To get rid of the older files of a part or sketch, use **MODE(Dbms) / DBMS(Purge)**. You can also use the 'purge' command from the unix shell.

Several of the functions mentioned in the guide do not appear in the standard Pro/ENGINEER format (menu picks such as Make_PART and Trim have been added to the menus in the manual). You can customize your Pro/ENGINEER system by editing your config.pro and menu_def.pro files. See Appendix A for instructions on editing your configuration files.

To start Pro/ENGINEER, enter the start-up command at a Unix prompt. Once Pro/ENGINEER is running, go to Section II to create your first part.

SECTION II

Creating a Simple Solid Part

Pro/ENGINEER starts up with a large, empty, blue window, two small menu windows and a message window. When you need to enter information with the keyboard, such as the part name or dimension values, it will be at a prompt in the message window. The message window will also often tell you what your next task is. Keep your eye on this box. There are times when you cannot continue until you enter a piece of information here.
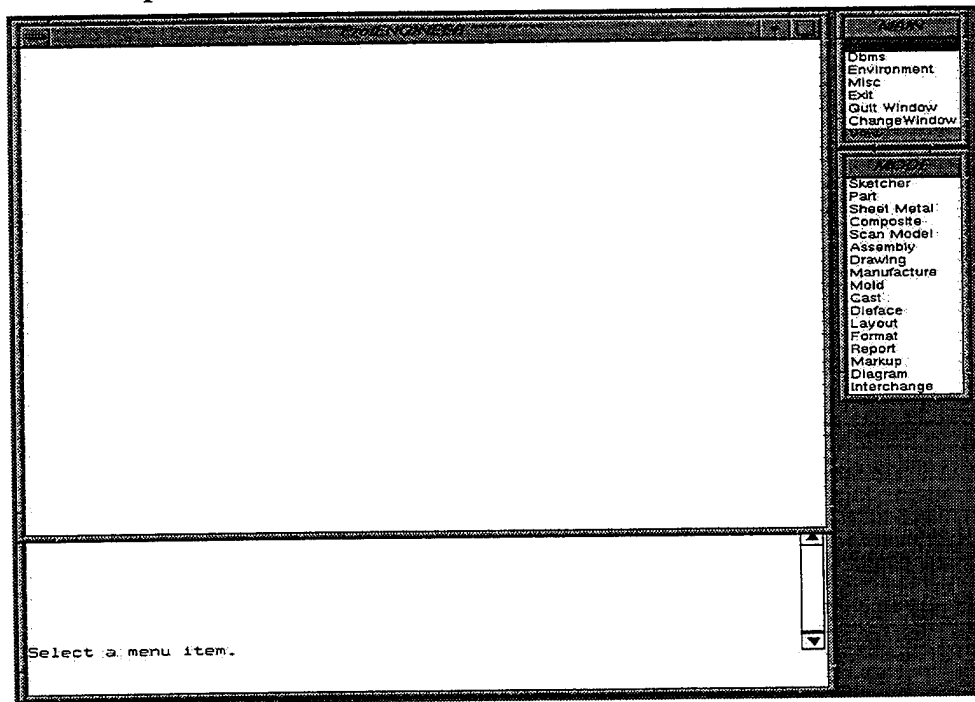

Figure (1) Pro/ENGINEER start-up display.

To start a part, use the following commands.[1]

MAIN(Mode) / MODE(Part) / ENTERPART(Create)

3

At this point, you must enter the name of your part at the "Enter part name [PRT0001]:" prompt in the message window.

MSG: test_solid <enter>[1]

You are now working on a .prt (part) file called test_solid.

First you will create the datum planes that your part will be aligned to.[2]

PART(Feature) / FEATURE(Create) / FEAT_CLASS(Datum) / DATUM(Plane) / MENUDTM_OPT(Default)[3]

A set of three perpendicular planes appear on the main window. It may be hard to tell that this is what you are looking at at first. Datum planes have a positive (yellow) and a negative (red) side.

The next feature you will create is a solid protrusion. Eventually you will have a rectangular solid box.

FEATURE(Create) / FEAT_CLASS(Solid) / SOLID(Protrusion) / FORM(Extrude) / FORM(Solid) / FORM(Done) / SIDES(One_Side) / SPEC_TO(Blind) / SPEC_TO(Done)[4]

After making these selections, the message window prompts you to pick a sketching plane. Click on one of the datum labels.

<LMB> DTM1[5]

---

1. Throughout this report, all actions in **boldface** must be carried out by the Pro/ENGINEER user. When multiple menu selections have to be made, they will be seperated by a slash. The word in all caps is the MENU TITLE as it appears in Pro/ENGINEER and the word in (Parentheses) is the choice to make from that menu.

1. Information that must be entered at the message window prompt will be *italicized* and prefaced with MSG: In this case, for example, just type test_solid and then hit enter at the prompt. You must always hit enter after inputting information at the message window prompt, however from here on, <enter> will be left out after the information to enter is given.

2. This is a set of coordinate planes that your part will be built up around. For all practical purposes, they are neccessary for all parts.

3. When making several choices to define a feature, as you will usually have to do, there may be several menu windows on the screen at once. Click the desired choice from each menu window. When all the menu windows have the desired option highlighted in red, click Done and you will proceed to the next step. Reference pg 1-7 of Pro/ENGINEER Part Modeling User's Guide.

4. If you make an incorrect choice, such as picking the wrong menu item here, you can always pick Quit from the Quit/Done menu and then CONFIMATION(Confirm) . This will clear all of your choices and send you back to the FEAT menu where you can restart your feature. You may also be able to pick Unselect or Unsel Last from the GET_SELECT menu. If you make a wrong menu choice, you can often return to the previous menu by picking Done/Return.

Note the red arrow on the corner of the DTM1 plane, which shows the direction you will be looking. For now it will not matter which side of the plane you are sketching on.

DIRECTION(Okay)

Next you will pick an orientation for your sketching plane. When orienting a sketch, you will choose a datum plane and a placement on the drawing window for that plane. The orientation plane must be perpendicular to the sketching plane. In this exercise, you will put the yellow, or positive, side of the DTM2 plane at the bottom of the screen.

SKET_VIEW(Bottom) / SETUP_PLANE(Plane) / GET_SELECT(Pick)

<LMB> DTM2

Now you are in the sketcher mode.[1] You should see a rectangular grid on the drawing window. Here you will sketch onto DTM1 the cross section that will eventually be extruded. Using the mouse and the sketching menu, you will be able to sketch lines, circles and arcs.[2] Don't worry about being exact, if you roughly sketch a rectangle, Pro/ENGINEER will know that it is supposed to be a rectangle. Dimensions will be added later and will not depend on the shape of rectangle you sketch.

SKETCHER(Sketch) / GEOMETRY(Mouse_Sketch)

For this exercise, just **draw a rectangle** centered on the origin[3].

<LMB> on a starting point.

You will see a red line from the starting point to the current position of your mouse pointer.

---

5. Picks from the drawing window are prefaced by <LMB> for Left Mouse Button (or <MMB> or <RMB> for the middle and right mouse button). Here the user should pick on DTM1 with the left mouse button.

1. Reference Pro/ENGINEER Parts User's Guide section 2.

2. Reference Pro/ENGINEER Part Modeling User's Guide pages 2-6 to 2-12.

3. Make sure that width of the rectangle is clearly different from the height to differentiate it from a square. The actual aspect ratio will be determined later.

Drag the mouse to create a horizontal line.

<LMB> to place the second corner of your rectangle and start a vertical line.

Continue this process until the box is complete.
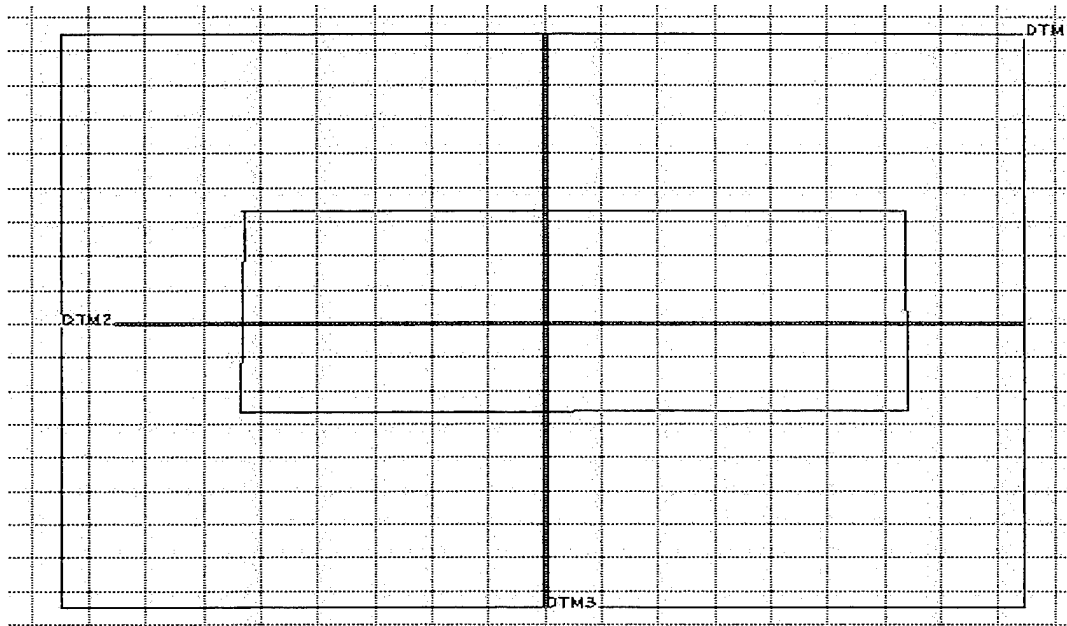
<MMB> anywhere on the main window.

Figure (2) A rectangle drawn in the Sketcher.

When you have completed the rectangle you will need to dimension it.[1] You need just enough information to completely define your cross section.[2] The cross section also needs to be dimensioned to the datum planes so that Pro/ENGI-NEER (and you) know where it is located in space. So a dimension must be added from DTM2 and DTM3 to the sketch.

Dimensions are placed by first using the <LMB> to pick the two edges that you want the distance between and then placing the dimension with the <MMB>.

SKETCHER(Dimension) / DIMENSION(Normal) / GET_SELECT(Pick)

<LMB> the left edge of your rectangle.

---

1. Reference Pro/ENGINEER Part Modeling User's Guide pages 2-19 to 2-23.
2. A square needs only one dimension, a rectangle needs two, etc.

You will see the edge change from light blue to red.

<LMB> the right edge of the rectangle.

Again, the edge will change color.

<MMB> once above the rectangle.

<MMB> places the dimension between the highlighted entities. Reference lines will extend from the left and right edges of the rectangle and a dimension labeled "sd#" will be placed between the reference lines at the spot where your mouse pointer is when you <MMB>.

Use the same procedure to dimension the height of the box (see Figure 3 for the dimension placements).

Now you must position the cross section with respect to the datum planes.

Place a dimension between the top of the rectangle and the horizontal datum plane.[1]

Place a dimension between the left side of the rectangle and the vertical datum plane.

Now you have the four dimensions you need to define and place the rectangle.

The next step is to regenerate the sketch.[2]

SKETCHER(Regenerate)

You will get a message at the bottom of the screen that says "Regeneration Completed Successfully." For some reason the regeneration may not work. Try to use the error message from the message window to figure out what needs to be dimensioned, or if some of your edges need to be trimmed. Add or delete[3] dimensions or trim edges[4] until the cross section regenerates without a problem

---

1. When sketching on DTM1, DTM2 and DTM3 will appear as horizontal and vertical lines.

2. Regenerating sketches or parts is a common step throughout Pro/ENGINEER before something is created or changed. It allows Pro/ENGINEER to completely rebuild the part, making sure that everything is dimensioned and aligned. Unsuccessful regenerations are freqeunt, one missing dimension or alignment will not allow the part to regenerate.

3. Pick SKETCHER(Delete) and then click on the unwanted dimension.

4. To trim edge lines that extend too far from the box, select Trim from the SKETCHER or GEOM TOOLS menu. You will be asked to select the line to trim to first and the line you are trimming second. Be sure to click on the line to be trimmed, not the part of that line that you want trimmed off. Reference Pro/ENGINEER Part Modeling User's Guide page 2-16 to 2-17.

(after each change, pick SKETCHER(Regenerate) to see if the object is now properly dimensioned). When the regeneration is successful, you will be automatically sent into the Modify mode.

Once a dimension is in place, you can change its value to whatever you want. To change the horizontal dimension of the box for example, use the following commands.

SKETCHER(Modify)

<LMB> the horizontal dimension label.

MSG: 6[1]

When you hit enter, you will see the label change to "6.00." Alter the other dimensions using this procedure.

Make the vertical dimension of the box *4*.

Make the distance from the horizontal datum to the top of the box *2*.

Make the distance from the vertical datum to the side of the box *3*.

SKETCHER(Regenerate)

---

1. This dimension has no units such as inches or centimeters. This becomes relevant later, when calculating mass or manufacturing material.

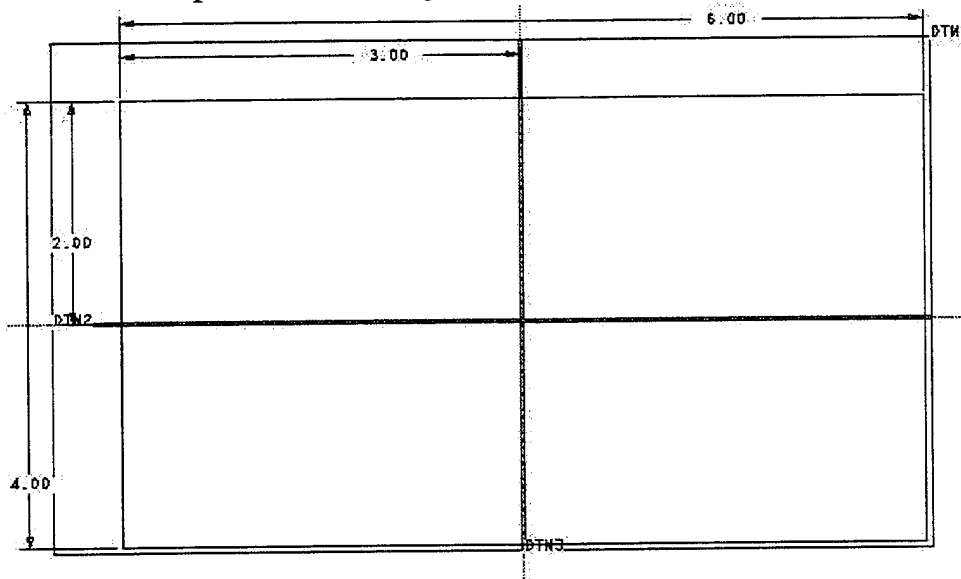You will see the shape of the rectangle redrawn to the new dimensions.

Figure (3) Finished rectangle, dimensioned and ready to extrude.

Once the regeneration is successful, you will see a clean drawing of your cross section. You are ready to extrude it.

SKETCHER(Done)

This means that you are finished with the sketcher and ready to extrude the cross section. Next, the message window prompts you for the depth of the extrusion.

MSG: 5

The extrusion has now been made and you are back to the FEAT menu. You will still see only a rectangle at this point, though. For proof that your solid exists, switch to the 'default' isometric view.

MAIN(View) / MAINVIEW(Orientation) / ORIENTATION(Default)

For a better idea of what you have you can create a 3-D shaded image of your box.

MAIN(View) / MAINVIEW(Cosmetic) / COSM_VIEW(Shade) / SHADE(Display)

To return to the unshaded view click view and repaint.

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MAIN(View) / MAINVIEW(Repaint)

Next you will cut a hole through the solid, your fifth feature (after the three datum planes and the protrusion).

FEAT(Create) / FEAT_CLASS(Solid) / SOLID(Hole) / OPTIONS(Linear) / OPTIONS(Done) / HOLE_FORM(Straight) / SIDES(One_Side) / SPEC_TO (Thru_All) / SPEC_TO (Done)

Next you will be asked for the diameter of the hole.

MSG: 1

After you enter this number, you will be asked for the placement plane that the hole will intersect.

<LMB> the top surface.

It should be easy to pick the desired surface from the default view. After you pick it, the edges of that surface will turn red. Next you will locate the axis of the hole.

<LMB> one edge of the selected surface.[1]

The edge will turn dark blue. Next, enter a distance at the message window prompt.

MSG: 2

<LMB> a second (adjacent) edge.

MSG: 2

The axis of the hole is now located, and the cut is made as soon as the second distance is entered.

As stated before, you can change any dimension on the part. Use the following commands to move the axis of the hole.

---

1. Try not to pick on a DTM plane.

FEAT(Done)

PART(Modify) / MODIFY(Value) / GET SELECT(Pick)

To modify a feature, <LMB> on it at this point. Here you will reposition the hole.

<LMB> somewhere on the hole.

The edges of the hole will turn red. The dimensions for the diameter and its distance from the edges of the part will show up on the screen.

<LMB> a dimension that locates the center.

MSG: 1

PART(Regenerate)

You will see the hole move to its new position.

To get a really good view of what you now have, you can spin a shaded view of your part.

MAIN(View) / MAINVIEW(Cosmetic) / COSM_VIEW(Shade) / SHADE(Display) / MAINVIEW(Orientation) / ORIENTATION(Spin)

<MMB> somewhere on your shaded view of the part.

Slowly move the pointer around to rotate the image.

<MMB> the part again to finish spinning.

When you are ready to quit, save the part.

MAIN(Dbms) / DBMS(Save)

MSG: <enter>

This step saves your part to the default name in brackets ([test_solid.prt]).
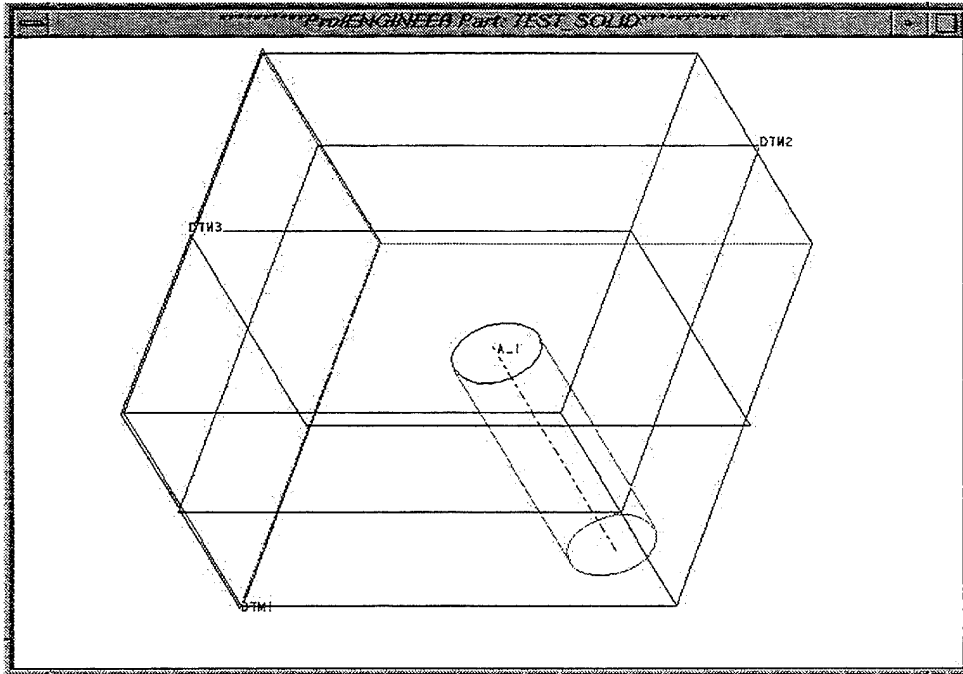
MAIN/(Exit)

<LMB> Yes

Figure (4) Final solid with datum planes.

SECTION III

Creating an Airfoil Section

Airfoil sections are 2D drawings that will be created in the sketcher. Before you start, you may want to make a directory for sections, or even one just for airfoil sections. Later, you will need to call up your airfoil section to make a wing surface. It may be easier to find and keep track of your sections if they are in their own directory. To start your section, use the following commands.

MAIN(Mode) / MODE(Sketcher) / ENTERSECTION(Create)

MSG: airfoil

The file will be a .sec (section) file.

The first step is to setup a coordinate system.
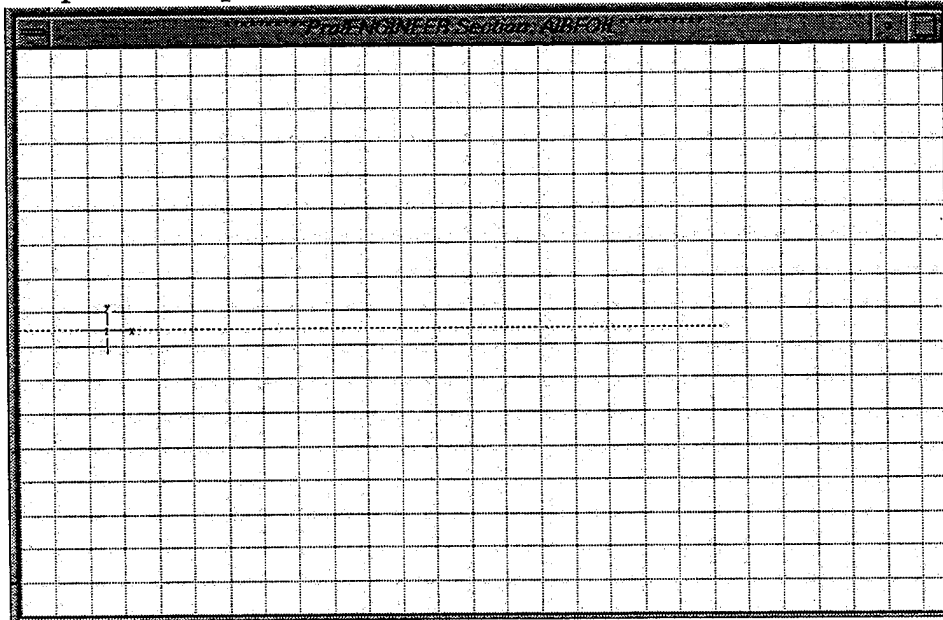
Figure(5) Sketcher setup with centerline and coordinate system.

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**SKETCHER(Sketch) / GEOMETRY(Adv_Geometry) /
ADV_GEOMETRY(Coord_Sys)**

Once these options are selected, you can place your coordinate system origin on the drawing window.

<LMB> near the left edge of the main window about halfway up (See Figure 5).

A small crosshair with axis labeled x and y and a z right in the middle will appear on the screen. The next step is to create a centerline along the x-axis.

**SKETCHER(Sketch) / GEOMETRY(Line) / LINE_TYPE(Centerline) /
LINE_TYPE(Horizontal)**

<LMB> on the x-axis (or right on the little z).

A dashed centerline[1] will show up across the screen. Now you are ready to draw your splines, the top and bottom edges of the airfoil section.[2]

**SKETCHER(Sketch) / GEOMETRY(Adv_Geometry) /
ADV_GEOMETRY(Spline) / SPLINE_MODE(Sketch_Points) / TAN-
GENCY(Start)**

Be sure Start is selected in the Tangency menu window. See Figure 6 for what the splines should look like.

<LMB> as close to the origin (right on top of the little z) as possible.

A red rubberband will now go from the origin to the mouse pointer.

<LMB> again near the top center of the drawing window.

Now the red line will bend through this second point so that the curved line starting at the origin still ends at your mouse pointer.

<LMB> once more on the centerline near the right side of the window.

You should see a hump shape across the drawing window above the centerline.

---

1. You will use centerlines in your sketches often. They exist only in the sketcher, to help you create the sketch properly.
2. Reference Pro/ENGINEER Part Modeling User's Guide pages 2-13 to 2-15.

<MMB> to finish the spline.

Using the same procedure, draw the bottom edge of your airfoil section.

It should roughly mirror the top section about the centerline. Be sure that the bottom spline starts and ends at the same points as the top spline.[1]
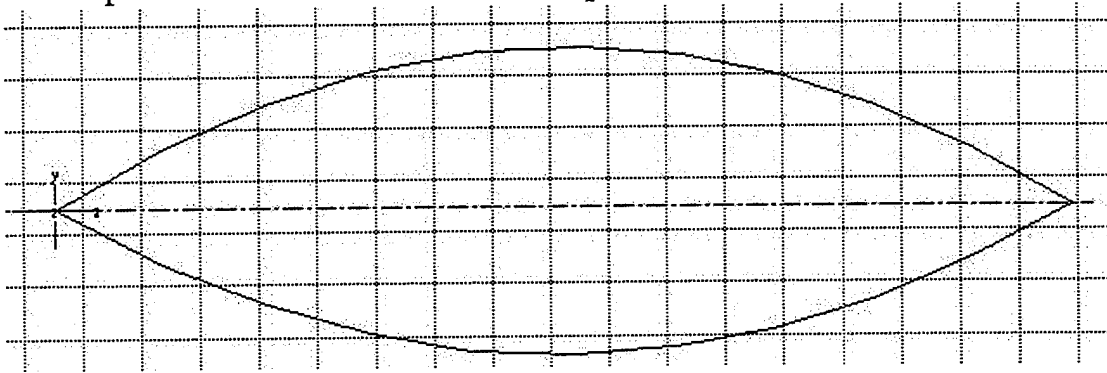
Figure(6) Unregenerated, undimensioned spline sketch.

The section is now drawn, but it must be dimensioned.

There are three steps to properly dimensioning an airfoil section: 1.) The Length 2.) The top and bottom leading angles (the angles between the splines and the centerline at the left vertex) 3.) The spline to the local coordinate system. It doesn't matter in which order these are done, but they must all be done. The latter two are somewhat tricky.[2]

SKETCHER(Dimension)

Length - Three steps:

1.) <LMB> the left vertex of airfoil.

2.) <LMB> the right vertex of the airfoil.

3.) <MMB> above the airfoil.

This places the dimension (high enough to be out of the way).

---

1. Reference Pro/ENGINEER Part Modeling User's Guide pages 2-38 to 2-40.
2. Reference Pro/ENGINEER Part Modeling User's Guide pages 2-23 to 2-24.

Leading angles - Five steps:

1.) <LMB> on either spline.

2.) <LMB> the SAME spline again (it will turn red).

3.) <LMB> the origin,[1] small, red crosshairs will show up on the end of the spline.

4.) <LMB> the centerline (somewhere inside the airfoil, it will turn red).

5.) <MMB> somewhere inside the angle.

Again, this places the dimension.

Repeat for the other leading angle.

Dimensioning to the local coordinate system - Seven Steps:

1.) <LMB> either spline.

2.) <LMB> the same spline again (it will turn red).

3.) GET SELECT(Query_Sel).[2]

4.) <LMB> the origin, ONLY ONE CLICK.

5.)Leaving the mouse pointer on the origin, <RMB> until the coordinate system symbol pops up in red (usually 2 or 3 clicks).[3]

6.) When the coordinate system is red, <MMB> anywhere on the screen.[4]

7.) <MMB> anywhere on the screen again to finish the dimension.

There is not an actual label placed when the splines are dimensioned to the local coordinate system, but you will get a message that says, "Spline is dimensioned to local coordinate system."

---

1. You are actually clicking on the endpoint of the spline, but they should be the same point.

2. You can also choose Query Select by clicking the RMB anywhere on the drawing window when the GET SELECT menu is visible.

3. Clicking the right mouse button in the Query Select mode moves through the possible features, or 'hidden geometry' to dimension to in the area under the mouse pointer.

4. This accepts the current choice from the Query Select.

Repeat for the other spline.

Now the object should be fully dimensioned.

SKETCHER(Regenerate)

Values for the length and angles will be entered by Pro/ENGINEER. Look at the message window, make sure there weren't any problems with the regeneration. If there were, follow the dimensioning instructions again very carefully.

SKETCHER(Modify)

Make the length of the airfoil 100.

Make the leading edge angles 90 degrees.

SKETCHER(Regenerate)
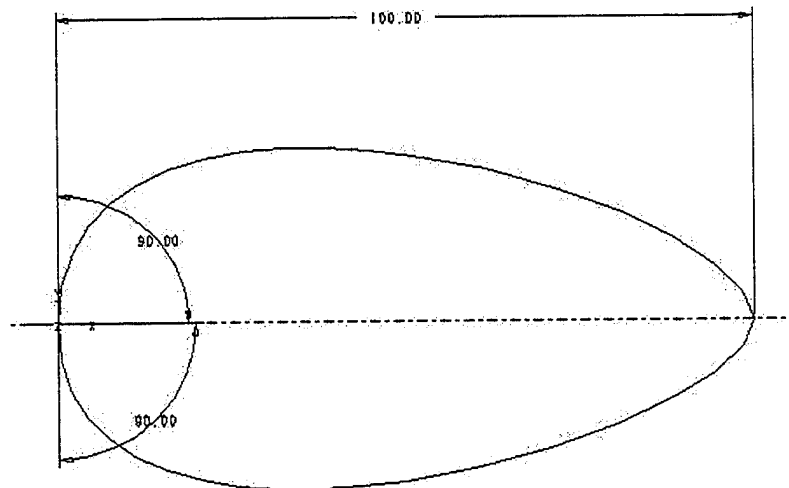
Figure(7) Dimensioned, regenerated section.

Your airfoil will now be more rounded and the grid lines from your coordinate system will be very close together[1] (because you have redimensioned the airfoil to be much larger). To clear up the grid lines, use the following commands.

---

1. It may be that there are no gridlines at all, in that case, they are too close together to display. The same procedure will fix the problem.

SKETCHER(Sec_Tools) / SEC_TOOLS(Grid) / MODIFY_GRID(Params) / CART_PARAMS(X&Y_Spacing)

Make the number of units between the grid lines 5 and repaint.

MSG: 5

MAIN(View) / MAINVIEW(Repaint)

This would be a good time to save your section drawing.

Main(Dbms) / DBMS(Save)

MSG: <enter>

Next you are going to use a set of (x,y) data points to define the shape of the airfoil[1].

SKETCHER(Modify)

<LMB> on the top edge of the airfoil.

Several menu windows will pop up in the bottom right corner of the screen. Bring the one on the bottom, called MOD SPLINE up to the top by clicking on the title bar.

MOD_SPLINE(Save_Pnts)

You will be asked to enter a file name for the data points that define the top edge of the wing, *MSG: airfoil_top*. Next you will be asked for the type of coordinates that will be used.

COORD_TYPE(Cartesian)

A file called airfoil_top.pts has now been created in your current directory.

Repeat the procedure to create a file called airfoil_bot for the bottom spline.

MAIN(Dbms) / DBMS(Save)

---

1. Reference Pro/ENGINEER Part Modeling User's Guide pages 2-38 to 2-40.

When you want to enter the data for your airfoil, close Pro/ENGINEER or put it in the background. Open the airfoil_top.pts file with a text editor. When you first open the file you will see a short message and then three columns for x, y, and z data.[1] The three points you used to define your spline will already be there. You will **enter your data points for the top edge of the airfoil into the x and y columns**, and **enter a z value of 0 for each data point.** The point (0,0,0) is the leading edge vertex of the airfoil, (100,0,0) is the trailing edge vertex. The x-axis is along the line between these vertices, and the y-axis is perpendicular to the ground. This is a 2D section, so all z values should be 0. **MULTIPLY ALL Y COORDINATE DATA POINTS BY NEGATIVE ONE.** Because of the way the coordinate systems are set up in Pro/ENGINEER, constructing your airfoil upside-down is the only way to make your wing end up right side up. When you are finished with the data entry, save and close the file. **Use the same procedure to enter the data for the bottom edge of the wing section into airfoil_bot.pts. Reenter Pro/ENGINEER.**

MODE(Sketcher) / SKETCH(Retreive)

MSG: airfoil

Now you will use the entered data points to define your airfoil.

SKETCHER(Modify)

<LMB> on the bottom spline.

MOD SPLINE(Read Pnts)

MSG: airfoil_top.pts.[2]

COORD TYPE(Cartesian)

You will get a message, 'File contains a different number of points. Do you want to continue? [Y]'

---

1. FIgure 2-23 from page 2-40 of the Pro/ENGINEER Part Modeling User's Guide.

2. Remember, because the y-data was multiplied by negative one, the 'top' points will now define the bottom spline.

MSG: <enter>

The default of yes is accepted. Your new spline should show up on the screen as a red line, and the data points will be red crosshairs.

SKETCHER(Regenerate)

You will see the old bottom edge disappear and the new one defined by the data points take its place.

Use the same procedure to define the top spline with airfoil_bot.pts.[1]

Save the section using the previous method and exit Pro/ENGINEER.
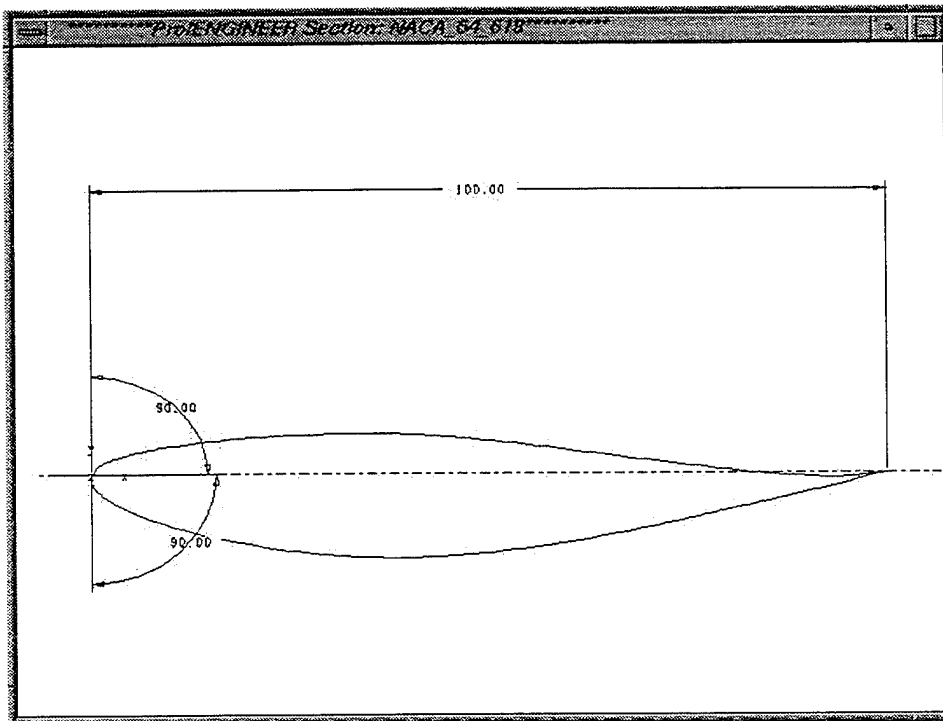
Figure(8) Completed wing section (exact shape will vary from wing to wing).

---

1. Zoom in very close on the leading edge of the airfoil. The front should be smooth and rounded. If it has a barb shape on the front, add 180 degrees to each of your leading edge angles. Regenerate. Then delete the angles and add them in again as 90 degrees.

SECTION IV

Creating a Wing Surface

The wing surface will be created by first creating "trajectories" that will define the leading and trailing edges of the wing. Your airfoil section from Section III will then be swept out along these trajectories to create the surface.

To start the wing surface part, you will have to **create a set of default datum planes and a default coordinate system**. It will be easiest to use the Make_PART[1] function to do this automatically.

   MAIN(Dbms) / DBMS(Make_PART)

You will see the datums and the coordinate system pop up on the screen and will be prompted to name the part. *MSG: wing*. You will be sent back to the original screen, but there will be a menu window from which you can pick **current_directory(wing.prt)**. Once you do this, you will see the datum planes and coordinate system origin (labeled 'CSO') again.[2]

Your next step will be to add two more datum planes.[3] One will contain the root chord of the wing. The other will contain the tip chord of the wing.

   **PART(Feature) / FEAT(Create) / FEAT_CLASS(Datum) / DATUM(Plane) /
   DATUM_PLANE(Offset)**

Your new datum plane will be offset from DTM3.

---

1. Pro/ENGINEER does not come with the Make_PART function, see Appendix A for directions on adding Make_PART to your config.pro file.

2. The x-axis is the lengthwise axis of the plane, it is perpendicular to DTM1. The y-axis is perpendicular to DTM3, the symmetry plane of your aircraft. The z-axis is perpendicular DTM2, the midsurface of your wing.

3. Reference Pro/ENGINEER Part Modeling User's Guide pages 3-1 to 3-2.

<LMB> DTM3

OFFSET(Enter_Value)

MSG: 20

Select **DATUM_PLANE(Done)** and you will see a plane labeled DTM4 show up parallel to DTM3.

DATUM_PLANE(Done) / FEAT(Done)

Repeat the process, offsetting a distance of 100 from DTM4.

To make things less cryptic, you will now rename DTM4 and DTM5.

PART(Set_Up) / PART_SETUP(Name) / NAME_SETUP(Feature)

<LMB> DTM4 and enter *MSG: wingroot.* **Name DTM5 wingtip**.

Next you will create three datum curves,[1] construction lines that will help place several other features as you build your wing. In essence, you are going to define the planform of the wing.

PART(Feature) / FEAT(Create) / FEAT_CLASS(Datum) / DATUM(Curve) / OPTIONS(Sketch) / OPTIONS(No_Xhatch) / OPTIONS(Done) / SETUP_PLANE(Plane)

You now need to choose your sketching plane.
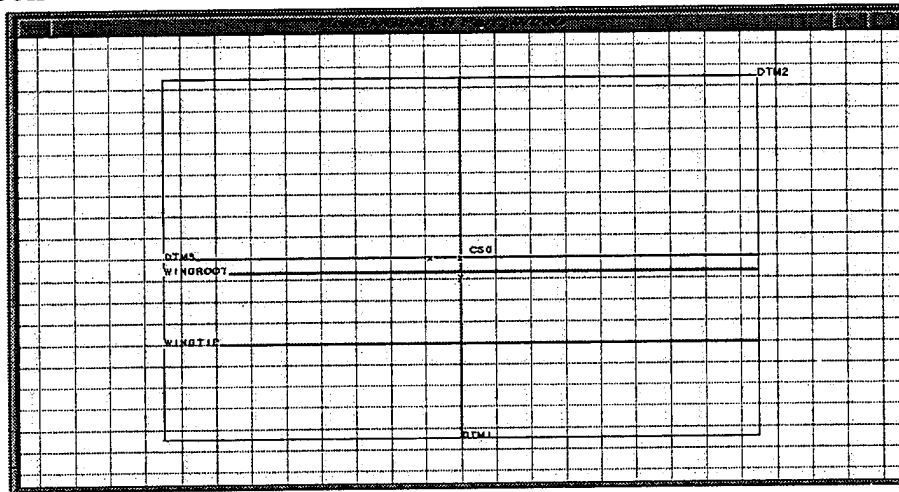
<LMB> DTM2

DTM2 is the equivalent of the wing midsurface. Picking DTM2 as your sketching plane means that you will be looking at your aircraft from above. To orient the sketch, pick DTM3.

<LMB> DTM3

---

1. Reference Pro/ENGINEER Part Modeling User's Guide pages 3-16 to 3-18.

Your screen will now look like this.

Figure(9) Sketching view.

You are now in the sketcher. Zoom in on the bottom left quadrant.

MAIN(View) / MAINVIEW(Pan/Zoom) / PAN-ZOOM(Zoom_In)

Draw a box around the lower left quadrant of the screen by placing the opposite corners of the zoom box with the LMB. DTM1 should be on the right side of the screen and the wingtip DTM should be on the bottom. (See Figure 10.)

MAINVIEW(Done-Return)

Draw three lines from Wingroot to Wingtip (see Figure 10). Pick GEOME-TRY(Mouse_Sketch), then <LMB> to start the line, <LMB> to end the line, <MMB> to finish.[1] These lines will be the leading and trailing edges of the wing and a construction line in the middle (this line will be used to define the sweep angle). The front of the aircraft is on the right from this view, so the lines that define the wing will slope back to the left. You will also have to add a centerline.

SKETCHER(Sketch) / GEOMETRY(Line) / LINE_TYPE(Centerline) / LINE_TYPE(Vertical)

---

1. Simple activities that have already been covered will no longer be explained in detail. If the user gets stuck, refer to previous sections.

This vertical line should be placed so that it intersects Wingroot and the middle line of the three you just created (dashed line on Figure 10).
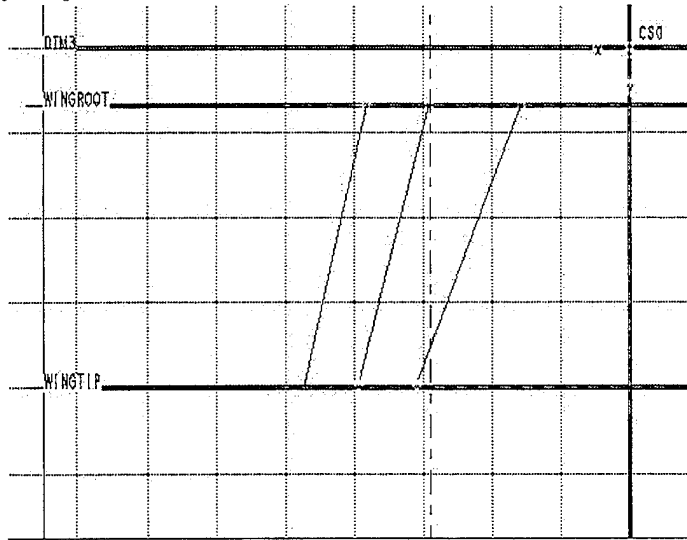
Figure(10) Construction line sketch.

For now, exaggerate the sweep angle of the wing. This will prevent Pro/ENGINEER from assuming that all of the lines are parallel to DTM1.

The endpoints of all three lines will have to be aligned to the Wingtip and Wingroot datums. Aligning an endpoint to a feature such as a datum tells Pro/ENGINEER that that endpoint will always lie on that datum.

SKETCHER(Alignment)

<LMB> the endpoint you want to align.

A red circle with small crosshairs through it will appear around the end of the line.

<LMB> the label of the datum plane that you want to align the endpoint to.[1]

If you did it right, you will see the message '--ALIGNED--' in the message window. Be sure to align both ends of each line to the corresponding datum (Wingroot or Wingtip).

---

1. You can also just <LMB> the datum plane line, so in most cases a double click on the end point of the line will align it to the correct datum plane, as long as you are in Alignment.

Now you must dimension the lines. Six dimensions will need to be placed:

1.) The distance from the DTM1 line to the top (wing root) point of the leading edge[1]

2,3.) The distance from the leading edge line to the trailing edge line on the wingroot and the wingtip

4,5.) The distance from the leading edge line to the middle line on the wingroot and wingtip

6.) The angle[2] between the centerline and the middle line

Now **regenerate** the drawing. If it does not regenerate correctly, make sure that all six endpoints are aligned and all six dimensions are included. **Modify the distance from leading edge to trailing edge along Wingroot to 100**. You can modify the other dimensions to round numbers if you want to. Pro/ENGINEER may automatically zoom in too close on your sketch after you regenerate. You can return to the previous view using Zoom Reset.

MAIN(View) / MAINVIEW(Pan/Zoom) / PAN-ZOOM(Reset)

You can also add relations to your dimensions, making certain dimensions functions of others.[3]

SKETCHER(Relation) / RELATIONS(Add)

Notice that all of the dimension labels are now numbered. This allows you to identify each dimension when adding relations. Enter *MSG: sdx=.25*sdy* where *sdx* is the dimension symbol for the distance from the leading edge to the middle line and *sdy* is the dimension symbol for the distance from the leading edge to the trailing edge along Wingroot. **Enter in a second, similar relation for the Wingtip. Hit enter again to quit adding relations. Regenerate** your drawing, and the numbers will change so that the middle line is always at the quarter-

---

1. Remember, Pro/ENGINEER needs to know where the drawing is in relation to the datums. You may not see the DTM1 label from this view, but you should know where it is, if not, zoom out.

2. Angular dimensions are constructed in three steps. <LMB> one side of the angle, <LMB> the other side of the angle, <MMB> to place the dimension.

3. Reference Pro/ENGINEER Fundamentals, section 15.

chord of the wing. Pick **Done** to finish the sketch. Pro/ENGINEER may zoom in too close on your drawing when you click done. To reset the view, choose **PAN-ZOOM(Reset)**. The light blue sketcher lines turn brown to show that they are now finished datum curves.

The planform has now been defined. The planform lines will not be used to help create the trajectories which the surface will be constructed on.

You now need to create three more datum curves, the trajectories that will be used to sweep out your surface. Each of these curves must be an individual feature.

> **PART(Feature) / FEAT(Create) / FEAT_CLASS(Datum) / DATUM(Curve) / OPTIONS(Sketch) / OPTIONS(No_Xhatch) / OPTIONS(Done)**

To save time and insure that you have the same view that you were using to sketch before, just click **PRE_SKETCH(Use_Prev)** when you are asked to setup the sketching plane (pick Okay if given an Okay/Flip menu). This will put you back into the same sketching view (looking at the aircraft from above, nose at the right, with your wing sloping from top right to bottom left).

The first of these new datum curves will be an exact copy of the leading edge line. The Use Edge command copies existing lines.

> **SKETCHER(Geom_Tools) / GEOM_TOOLS(Use_Edge)**
>
> **<LMB> the leading edge line.**

It will turn from brown to light blue. **Regenerate**, there shouldn't be a problem. **Pick Done until you get back to the FEAT menu**. This will insure that the datum curve you just created will be an individual feature, unlike the three construction lines that were all part of the same feature. Using the same procedure that you used to rename DTM4 and DTM5, **rename this datum curve 'trajectory_1.'**

> **Repeat the procedure to make a copy of the trailing edge line. Name it 'trajectory_2.'**

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The <u>third of the three new datum curves will be the spine</u> for the sweep that will eventually create your surface.

> **PART(Feature) / FEAT(Create) / FEAT_CLASS(Datum) / DATUM(Curve) / OPTIONS(Sketch) / OPTIONS(No_Xhatch) / OPTIONS(Done) / PRE_SKETCH(Use_Prev)**

These are the same steps that you used before to make the last two datum curves, but this time you are going to actually draw the line rather than copy an existing one. **Make a line along the DTM1 plane** (it should appear as a vertical line on the right side of the page) **from the point where Wingroot intersects to the point where Wingtip intersects.**[1] Unlike the last two datum curves, this one will have to be aligned (the other two were copies of lines which you previously aligned). First align the line to DTM1. Here you are aligning the <u>line</u> to be contained in the DTM1 plane. Before you were aligning endpoints.

> **SKETCHER(Alignment)**
>
> **<LMB> on the line (somewhere on the middle).**
>
> **<LMB> DTM1**

Align the endpoints to Wingtip and Wingroot using the method used for the construction lines. Click **Regenerate** and **Done. Name the datum curve 'spine.'**

All the lines that you need to make the surface are now in place. To make things more clear, you will remove the construction lines from the view, so that only the three trajectories are visible. Pro/ENGINEER has several functions to help you keep your display clear. The most basic is layering. Features are put in different 'layers', and those layers can be turned on or off. **Pick Done from the FEAT menu** to get back to the PART menu.[2]

---

1. The intersections may not be explicitly shown. The datum planes go to infinity in all directions but Pro/ENGINEER shows them as rectangles. Put the endpoints for the line where the wingtip and wingroot datums would intersect if they were extended.

2. Reference Pro/ENGINEER Fundamentals pages 11-1 to 11-5.

PART(Layer) / LAYERS(Setup_Layer) / SETUP_LAYER(Create)

MSG: construction

MSG: <enter>

Now you want to add your construction lines to your new Construction layer.

LAYERS(Set_Items) / LAYER_SEL(Construction)

Layers you create will be placed in the Layer Select menu. In this menu, items will be checked or unchecked when you click on them, this allows you to make multiple selections.

GET_SELECT(Done_Sel) / SET_LAYER(Add_Items) / LAYER_OBJ(Feature)

Now **<LMB> the middle line**, you will see three lines turn red, since the middle line was created in the same feature at the other two. Pick **Done Sel** to finish adding features to the Construction layer. Now you will turn off the Construction layer.

LAYER(Set_Display) / LAYER_SEL(Construction)

A check will appear by 'CONSTRUCTION.' This means that the Construction layer will not be visible anymore (unless you turn it back on). Pick **Done Sel** and **LAYERS(Repaint).**[1] When the image redraws, you will see that the middle line has now disappeared (the other two remain because you made individual copies). Table 1 shows your current layer information. Included are the four default layers setup by Pro/ENGINEER and your new layer, Construction. Also included is the status of each layer and the items contained in it.

Layer Name	Status	Items in Layer
Construction	Off	three construction datum lines
DEF_DTMS	On	default datums (DTM1, DTM2, and DTM3)
DTM_CSYS	On	coordinate system origin

TABLE (1) Current Layer Information

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1. If you did not add the lines from Appendix A to your menu_def.pro, Repaint will not be in your LAYERS menu. You will have to repaint from he MAINVIEW menu.

Layer Name	Status	Items in Layer
DTM_AXES	On	any datum axes you create, now empty
DTMS_PLNS	On	all non-default datum planes, now wingroot and wingtip

TABLE (1) Current Layer Information

Now it is time to create the surface.

> **PART(Feature) / FEAT(Create) / FEAT_CLASS(Surface) / QUILT_SURF(New) / SURF_FORM(Advanced, Done) / ADV_SRF_OPT(Var_Sec_Swp, Done) / OPTIONS(Nrm_to_Spine, Done) / VAR_SEC_SWP(Select_Traj) / DEFINE_CHAIN(Select) / CHAIN_TYPE(One_By_One)**

Var Sec Swp stands for Variable Section Sweep,[1] the type of surface you are creating. Nrm to Spine means that the surface will be swept out normal to the spine datum curve you made. Select Traj will allow you to select the sweep trajectories one by one. As your first trajectory, you must choose the spine. **<LMB> on the spine**. It will turn red. Since the spine curve is right along a datum plane in this view, you may want to use Query Select to pick it. Pick **Done Sel**, the red line will turn light blue and there will be an arrow at one end showing the direction of the sweep. Pick **Done** and then go on to the second trajectory.

> **VAR_SEC_SWP(Sel_Traj) / DEFINE_CHAIN(Select) / CHAIN_TYPE(One_by_One)**

This time **<LMB> the leading edge** and then pick **Done Sel**. It will turn light blue as well, but there will be no arrow. Click **Done** and **repeat the process with the trailing edge**. All three trajectories have now been selected.

> **VAR_SEC_SWP(Done) / SURF_END(Open_Ends) / SURF_END(Done)**

You are now put in the sketcher mode where you will sketch the cross section that will be used to sweep out your surface. You are given a view that shows the endpoints of your three trajectories as light blue Xs. The one on the far left is the spine. The cross section will be placed between the other two. Instead of sketch-

---

1. Reference Pro/ENGINEER Part Modeling User's Guide pages 4-54 to 4-65.

ing a new cross section here, you will use the airfoil section that you created in the last section.[1]

SKETCHER(Sec_Tools) / SEC_TOOLS(Retrieve_Section)

Enter *MSG: airfoil*[2] or just enter MSG: ?.[3] This will bring up a menu window from which you can pick the airfoil.[4] A small window with a picture of your airfoil in it will show up on the top right side of the screen. To avoid problems, follow the directions for positioning the section to be swept out very carefully. You will first be asked for the angle of rotation of the sketch.

MSG: <enter>

This accepts the default rotation angle of 0. Next you will pick the origin of scaling.

<LMB> on the front vertex of the airfoil in the small window.

Next you will be asked for the drag point.

<LMB> on the back vertex of the airfoil.

Now you will be asked for a scaling factor, accept the default.

MSG: <enter>

WITHOUT CLICKING ANY MOUSE BUTTONS, drag the section down onto the large screen.

It will show up in red.

Position the front vertex of the section on the middle blue X and <RMB>.

---

1. Reference Pro/ENGINEER Part Modeling User's Guide pages 4-5 to 4-7.

2. If your section has a different filename, be sure to enter it in all lower case.

3. If you know the name of the file, just type it in. On some displays, the graphics in the drawing window are effected when the new menu window pops up, making the Xs impossible to see. This makes it impossible to place the sketch. Resizing the small window that pops up or making other changes can effect the display as well. When retrieving the section, try to follow the steps listed and NONE OTHERS.

4. If it is in a different directory, you can move around your directory tree in the menu window to find your section.

This will freeze the front vertex to the middle light blue X (where you want it) and allow you to then scale the section so that the back vertex is on the far-right blue X (also where you want it).

Now position the mouse pointer (back vertex) on the right blue X and <LMB>.

The section is now positioned with each endpoint on one trajectory. The front and back vertex must be aligned to the trajectories. **From SKETCHER(Align), <LMB> on the front vertex of the wing section, then <LMB> on the X right under it.**[1] **Now do the same thing for the back vertex. Delete the chord dimension with SKETCHER(delete) <LMB> on chord dimension. SKETCHER(Regenerate)** the sketch.

If the regeneration was successful, pick **Done**. The surface will be created. To get an appreciable view of your wing surface, shade a default view.

MAIN(View) / MAINVIEW(Orientation) / ORIENTATION(Default) / MAIN-VIEW(Cosmetic) / COSM_VIEW(Shade) / SHADE(Display)
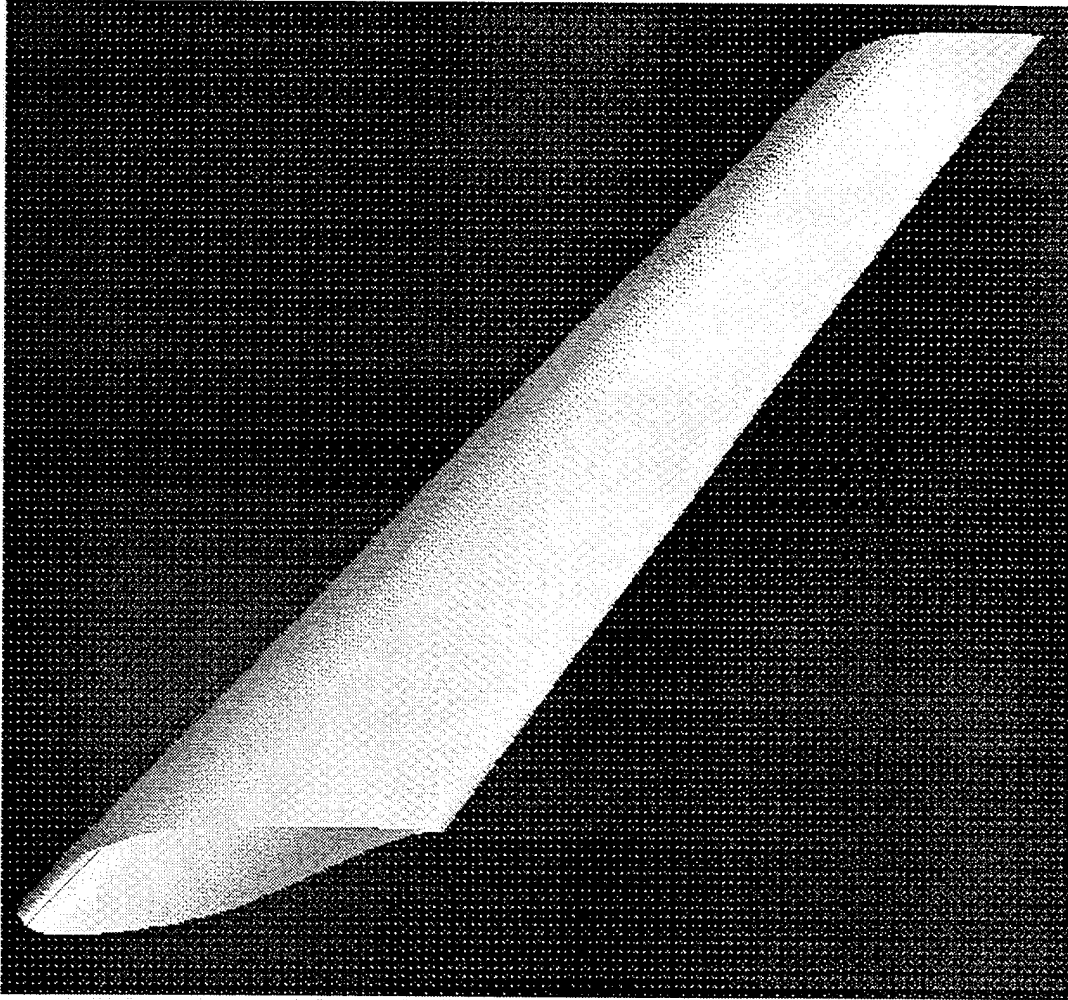
If you spin the view, you should be able to see through the length of the wing, since the surface was created with open ends.

Save the part, it will be used again in future sections.

---

1. They should be the same point.

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Exit Pro/ENGINEER.

Figure(11) Finished wing surface, shaded view.

SECTION V

Adding Rib and Spar Surfaces to a Wing

Now you are getting into more complex objects. You will be adding ribs and spars to your wing, using the wing surface you have created to define them. Things will become very confusing as more and more datum planes, datum curves, and surfaces are added to your part. It will become very important to have many well ordered layers so that unneeded features can be removed from the view. Otherwise, it will be impossible to clearly see the features that you need.

First, retrieve wing.prt.

> **MODE(Part) / ENTERPART(Retrieve)**

> *MSG: wing*

After arranging your display a little, you will make a mesh of datum curves to define the wingbox. The rib and spar surfaces, and eventually the rib and spar solids, will be places on these datum curves. Datum planes will be constructed through the lines of the wingbox and surfaces will be created on these planes.

For now you will need your construction lines but it will be easier if the surface feature and the trajectory lines are out of the way. **Change your view to the default** (it may be that way already).

> **PART(Layers) / LAYER(Setup Layer) / SETUP_LAYER(Create)**

Make two new layers.

> *MSG: trajectory*

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MSG: surface

MSG: <enter>

Under LAYERS(Set Display), turn the Construction layer off[1] and repaint.
It was off last time you were working on wing.prt, but Pro/ENGINEER opens a
part with all layers on. Now go to **Set Items**. You will see a menu window with
all of your layers in it; pick the layer **Surface** and then **Done Sel**. Then pick **Add
Items**. To put your surface in the Surface layer, **<LMB> the surface**. Pick **Done
Sel**.

Use the same procedure to put the three trajectory lines in the Trajectory layer.

Now pick **Set Display**. You will see the menu with all of your layers. **Pick Tra-
jectory and Surface** to turn those layers off. **Pick Construction** to turn it on
again (the check will disappear). **Repaint** to get an updated view of what you
have visible. You should only see your datum planes and the three construction
lines.

The first step to making rib and spar surfaces is to create a set of datum curves to
define the location of the wing box. Your rib and spar surfaces, as well as solids,
will be placed by this mesh of datum curves. All together, you will draw eight
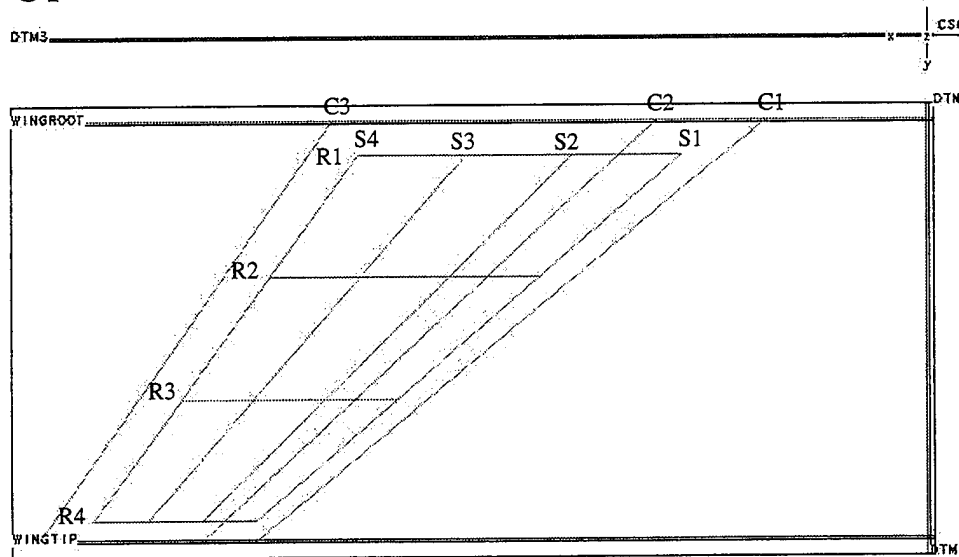lines, four for ribs and four for spars.

**PART(Feature) / FEAT(Create) / FEAT_CLASS(Datum) / DATUM(Curve) /
OPTIONS(Sketch, No_Xhatch, Done)**

---

1. This is just to make is easier to setup the trajectory layer.

Use the same method as before to setup the sketching plane, **pick DTM2 for the sketching plane and Bottom/DTM3 to orient the sketch.**

Figure(12) Sketch for wingbox datum curves.[1]

> **Use the Line/Horizontal function to draw lines to define the first and last rib.** Place them close to, but not on, Wingtip and Wingroot (labeled R1 and R4).

> **Draw four lines from R1 to R4 (labeled S1-S4 on Figure 12),** roughly evenly spaced between the leading and trailing edges. These lines will define the spars.

> **Draw two more horizontal lines to define the middle ribs** (labeled R2 and R3) roughly evenly spaced between R1 and R4.

Align and dimension these new lines. You should be starting to get the feel of what needs to be dimensioned and aligned. Remember that endpoints need to be aligned. Dimensions must be added so that the location of all objects relative to each other and the datum planes is known. Add dimensions in a meaningful way. Later, you may want to add relations. (S1 to S2, S1 to S3 and S1 to S4 along R1 and R4; R1 to R2, R1 to R3, R1 to R4; R1 to Wingroot; C1 to S1 at both ends (horizontal distance)) If you regenerate unsuccessfully, Pro/ENGINEER will tell you what the problem was and highlight problem dimensions or endpoints in red. When you **regenerate** successfully, pick **Done** to finish the feature. These

---

1. C = Existing construction line, R = Ribs, S = Spars. (These labels do not appear on the pro/ENGINEER display.)

lines will be part of one feature called 'boxmesh.' **Name it and put it in its own layer**[1] (also called boxmesh). Change your display so that the **Construction layer is turned off. Save.**

Your next step is to create eight, new, individual datum planes. This means that you will want to go all the way back to the FEAT menu after you make each one. Four of the planes will go through the four spars and four will go through the four ribs. All of these planes will be perpendicular to the wing midsurface, DTM2.

> **FEAT(Create) / FEAT_CLASS(Datum) / DATUM(Plane) /**
> **DATUM_PLANE(Through)**

<LBM> one of the rib or spar lines in the boxmesh (it doesn't matter which one, you will have to do them all, but it will be easier if you have an order to it). Your new datum is not defined yet, you have to make it normal to the wing midsurface. **Select DATUM_PLANE(Normal) and then <LMB> DTM2.** You should get a message saying that the plane is fully constrained.

> **Pick DATUM_PLANE(Done) twice to finish the feature as an individual entity.**

> **Repeat this process until you have all eight planes (R1-4 and S1-4 from Figure 11).**

> **Name your eight new datum planes Rib1-Rib4 (starting from the fuselage) and Spar1-Spar4 (starting from the nose). Save.**

Now you will make datum curves in these new planes to describe the boundaries of the spar and rib surfaces. Again, make sure that each one is created as a separate entity. **Go into layers and turn Boxmesh off and Surface back on.**

Layer Name	Status	Items in Layer
Construction	Off	3 construction datum lines
Trajectory	Off	3 trajectories

TABLE (2) Current Layer Information - Default layers on

---

1. The chain of menu choices connecting unrelated, but chronological steps will no longer be explicitly stated. The user will soon develop a feel for moving through the Pro/ENGINEER menus.

Layer Name	Status	Items in Layer
Surface	On	wing surface
Boxmesh	Off	wingbox datum curve mesh

TABLE (2) Current Layer Information - Default layers on

These eight datum curves will be formed by the intersections of the wing surface and the eight datum planes you just created.[1]

> **FEAT(Create) / FEAT_CLASS(Datum) / DATUM(Curve) / OPTIONS(Intr._ Srfs, Done) / INTR_SURFS(Single)**

<LBM> the Rib1 DTM. You will be returned to the INTR_SURFS menu. This time **pick INTR_SURFS(Whole)** and then **<LBM> the wing surface.** If you don't see the wing surface, you didn't repaint after turning the Surface layer back on. Get in the habit of repainting whenever you change the layer display. The datum curve is now fully defined and created. **Look at the default view to see a brown wing section now contained in the Rib1 datum plane**. That is the datum curve you just created.

> **Make eight new layers. Label them Rib1-4 and Spar1-4. Turn them all off right now.**

> **Put the Rib1 DTM and the datum curve you just made in the Rib1 layer.**

Now **create your Rib2 datum curve, using the same method as you did for Rib1**. When you are done, again **put the Rib2 DTM and the datum curve you made with it into the corresponding layer. Repeat for Rib3 and Rib4 and Spars 1-4. Be sure not to leave any steps out**. As you make each datum curve, and put it and the corresponding datum plane into inactive layers, your display will become more and more clear. (Note, when each curve is finished, it should be brown. Each spar curve will be a set of two almost parallel lines down the length of the wing.) At this point you can turn off all layers except for Rib1-4 and Spar1-4. You will see a brown 'wireframe' view of your wing made from datum curves.

---

1. Reference Pro/ENGINEER Part Modeling User's Guide page 3-18.

One more set of features must be added before the rib and spar surfaces can be created. These will be axes that define the ends of the of the surfaces. There should be one axis at each endpoint of the lines that make up boxmesh. Axes will be created at the intersections of datum planes.[1]

Feature / Create / Datum / DATUM(Axis) / DATUM_AXIS(Two_Planes)[2]

<LMB> the Rib1 and Spar1 labels. This will create an axis (dashed line labeled A_#) at the intersection of the Rib1 and Spar1 planes (the DTM_AXES layer was left on earlier so that you would be able to see this axis). **Pick Done. Repeat the process with Rib1 and Spar4**. These two axes will define the edges of your Rib1 surface. **Add six more axes to define the ends of the other three ribs**. Finally, **add four more axes to define the ends of Spar2 and Spar3** (the axes needed to define the ends of Spar1 and Spar4 are at the same as those that define Rib1 and Rib4). You should now have 12 axes on the outer edge of boxmesh. Take note of which axes are where. Put the axes at the ends of each rib into the corresponding rib layer. Put the axes at the ends of Spar2 and Spar3 in the corresponding layer. The Rib1 and Rib4 layers will have to be on when you create the Spar1 and Spar4 surfaces.

Now you can make the rib and spar surfaces. Start with the spar surfaces, they are slightly easier. At this point, **turn off all layers except for Spar1, Rib1, Rib4, DTM_PLNS and DTM_AXES**. Be sure you can see the two axes that define the ends of Spar1 (Int. of Spar1 with Rib1 and Rib4). Take note of the numbers of those axis. Now you will make the Spar1 surface.[3]

Feature / Create / Surface / New / SURF_FORM(Flat)

Your Sketching plane will be Spar1. To orient the sketch, pick **Top and DTM2 from Sel by Menu**, since it is not on the screen to pick.

---

1. Reference Pro/ENGINEER Part Modeling User's Guide pages 3-9 to 3-11.

2. For familiar menu commands, the MENU TITLE will no longer be displayed. Make the given choices from the menus on the screen.

3. Reference Pro/ENGINEER Part Modeling User's Guide page 6-5.

**GET_SELECT(Sel_by_Menu) / SELN_OPTIONS(Datum) /
SPECIFY_BY(Name) / NAMES(DTM2)**

You should now see a flat rectangle on the screen (the Spar1 DTM) with two brown lines in it.[1] The axes that will define the ends of your surface should be visible. Other axes, depending on your layer setup, may be visible as well.
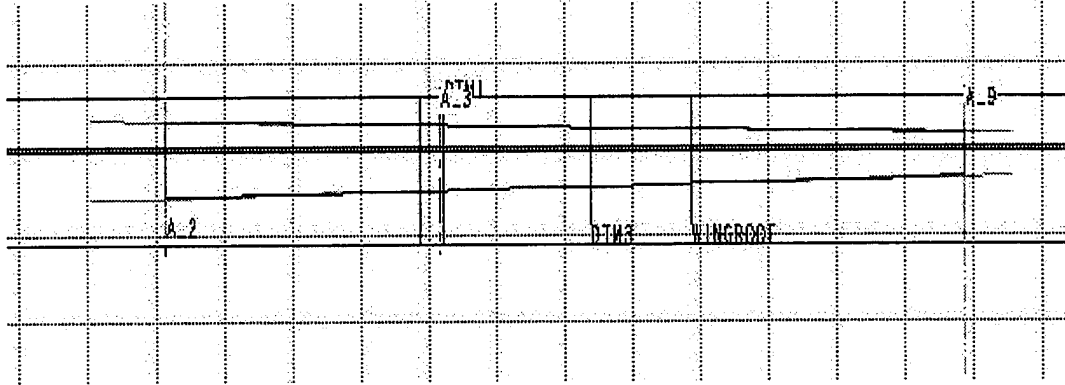
Figure(13) Completed spar surface sketch in solid line.

The sketch you want to draw is a box that has two of its edges on the brown lines and two edges on the end axes.

Draw lines on the axes, between the brown lines.

Align these lines to the axes they are on.

Use the Use Edge command to copy the brown lines.

Trim[2] the copied edges so that they extend only to the end axes.

Select Bound and then <LMB> the right axis line. Then <LMB> the top edge of the spar curve (on the part you want to keep, not the part you want to trim off). You will see the part of the curve to the right of the axis line turn brown again.[3] Now <LMB> the bottom line, the same thing will happen. Repeat the procedure for the left side of the curve.

---

1. There may be a few other lines from other datum planes. The brown lines may take up only a small portion of the view. If so, zoom in to do your sketch.

2. If you added the lines listed in Appendix A to your config.pro, Trim will be in the SKETCHER menu. If you didn't you can find it under SKETCHER(Geom Tools).

3. The blue sketcher line has been trimmed off to reveal the brown datum line again.

The sketch should now **regenerate** without a problem, if not, try aligning the endpoints. Pick **Done** to finish the surface. To see the surface you can shade it and then spin it around.

Create two more layers, Ribsurf and Sparsurf, turn them both off.

Put the surface you just created into Sparsurf.

Change your display so that Spar1 is off again and Spar2 is on.

Use the previous procedure to create the Spar2 surface. Be sure that the Spar2 DTM is your sketching plane.

When finished, put the surface in the Sparsurf layer.

Right now, your layers should be setup like this (not including the default layers). Try turning off all layers and then randomly turning them back on one at a time to make sure each feature is in the correct layer.[1]

Layer Name	Status	Items in Layer
Construction	Off	3 construction datum lines
Trajectory	Off	3 trajectories
Surface	Off	wing surface
Rib1	Off	Rib1 datum plane and curve, 2 axes
Rib2	Off	Rib2 datum plane and curve, 2 axes
Rib3	Off	Rib3 datum plane and curve, 2 axes
Rib4	Off	Rib4 datum plane and curve, 2 axes
Spar1	Off	Spar1 datum plane and curve
Spar2	On	Spar2 datum plane and curve, 2 axes
Spar3	Off	Spar3 datum plane and curve, 2 axes
Spar4	Off	Spar4 datum plane and curve
Ribsurf	Off	empty
Sparsurf	Off	2 spar surfaces
Boxmesh	Off	wingbox datum curve mesh

TABLE (3) Current Layer Information

Repeat for Spar3 and Spar4. Save.

---

1. Note that when a feature is in more than one layer, such as the rib and spar datum planes which are automatically put in the DTM_PLNS layer as well as the rib and spar layers you put them in, it will not show up on the display as long as any of the layers it is in are off.

Now you will make the rib surfaces. The surfaces will be contained in the rib datum planes. Their edges will be defined above and below by the top and bottom edges of the rib datum curve. The ends will be defined by the intersections of Spar1 and Spar4 (axes should be in place to sketch on).

Turn on Spar1, Spar4 and Rib1.

Turn off the Spar1 and Spar4 layers and repaint. Now create the surface.

Feature / Create / Surface / New / Flat

Pick Rib1 for the sketching plane and Top/DTM2 for the orientation.
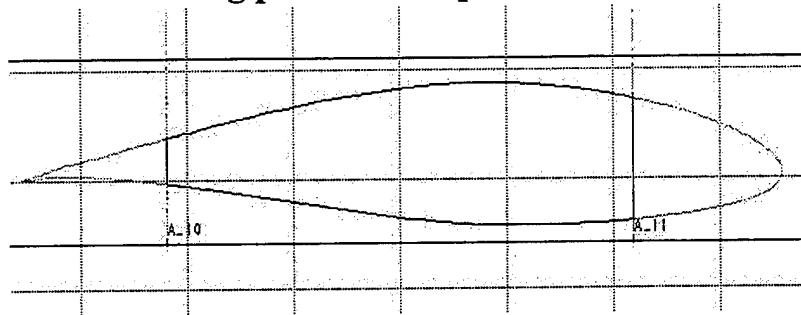
Figure(14) Rib surface sketching plane, completed sketch in solid line.

Draw lines from the top of the curve to the bottom of the curve along each axis.

Align each one to the axis it is on.

Use the Use Edge function to copy the top and bottom of the rib curve.

Use Trim to trim the copied curves so that they go only to the axis.

Regenerate the sketch, everything should be aligned. Pick **Done** to create the surface. You can view the surface using any of the discussed methods.

Put the surface in the Ribsurf layer, it needs to be out of the way.

Use the previous method to make your second rib surface.

Layer Name	Status	Items in Layer
Construction	Off	3 construction datum lines
Trajectory	Off	3 trajectories
Surface	Off	wing surface

TABLE (4) Current Layer Information

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Layer Name	Status	Items in Layer
Rib1	Off	Rib1 datum plane and curve, 2 axes
Rib2	On	Rib2 datum plane and curve, 2 axes
Rib3	Off	Rib3 datum plane and curve, 2 axes
Rib4	Off	Rib4 datum plane and curve, 2 axes
Spar1	On	Spar1 datum plane and curve
Spar2	Off	Spar2 datum plane and curve, 2 axes
Spar3	Off	Spar3 datum plane and curve, 2 axes
Spar4	Off	Spar4 datum plane and curve
Ribsurf	Off	2 rib surfaces
Sparsurf	Off	4 spar surfaces
Boxmesh	Off	wingbox datum curve mesh

TABLE (4) Current Layer Information

Repeat the procedure for the third and fourth rib surfaces. Remember to put items out of way in clearly ordered layers as you create them.

When you are finished, turn off all layers except Ribsurf and Sparsurf. Shade the resultant display and spin it around. Your four spar and four rib surfaces should form the internal structure for a wing.

Save and exit
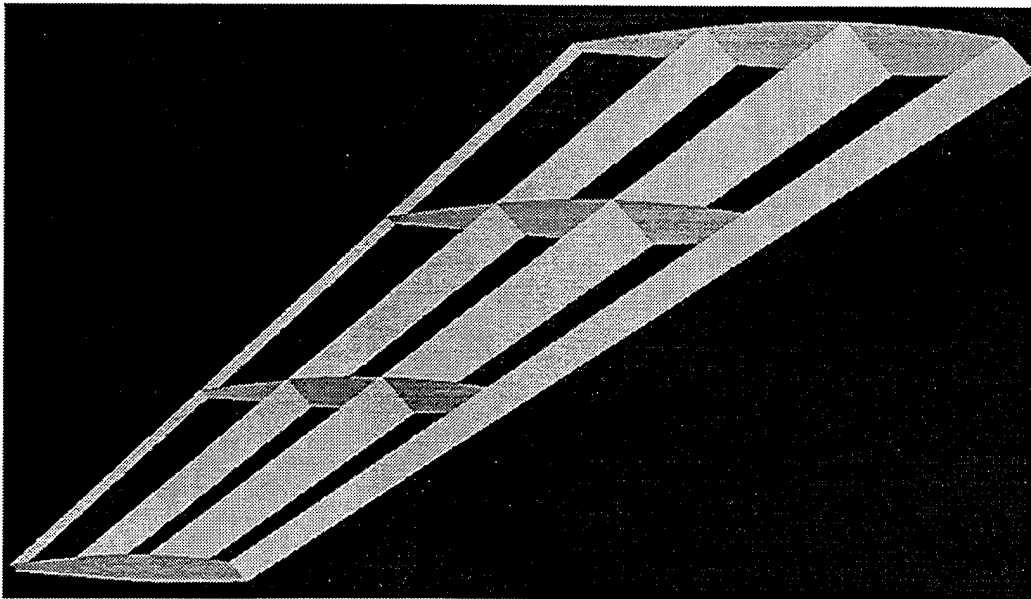
Figure(15) Completed spar surfaces, shaded view.

SECTION VI

Creating a solid wing skin

You now have a wing with an outer surface and rib and spar surfaces. In this section you will create a solid wing skin and in the next section you will make solid ribs and spars.

Open wing.prt.

Four datum curves will be required for each of the top and bottom skins. Two datum points will have to be created for each of the datum curves. **Turn off all layers except Rib1, Surface, Spar2, Spar3 and DTM_PLNS.**

> **PART(FEATURE) / FEAT(Create) / FEAT_CLASS(Datum) / DATUM (Point) / DATUM_POINT(Three_Srf)**
>
> **<LMB> on the Spar2 tag -the plane will be outlined in blue**
>
> **<LMB> on the Rib1 tag -the plane will be outlined in blue**

<LMB> on the top wing surface. Use Query Select to be sure that you pick the right surface. You should see an X mark with the label PNT0 appear at the intersection of Spar2, Rib1, and the top of the wing surface.

> **DTM_PNT_MODE(Done) / DATUM_POINT(Three_SRF) - Create another point at the intersectin of Spar3, Rib1 and the top wing surface.**

Turn off all layers except Rib1 and DTM_AXES. If you are not in a default view, switch to one now. You should see only the brown Rib1 wing section shaped datum curve, two rib axes, and the two datum points you just made.

Now create the datum curve.

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Feature / Create / Datum / Curve / Sketch, No Xhatch, Done

You will have to **use Sel By Menu to pick Rib1 as your sketching plane**. Use Sel by Menu again to **pick Top and DTM2 for the direction**.

Zoom in on the brown wing section. Look at the figure of the completed sketch and then follow the directions to draw and dimension it.
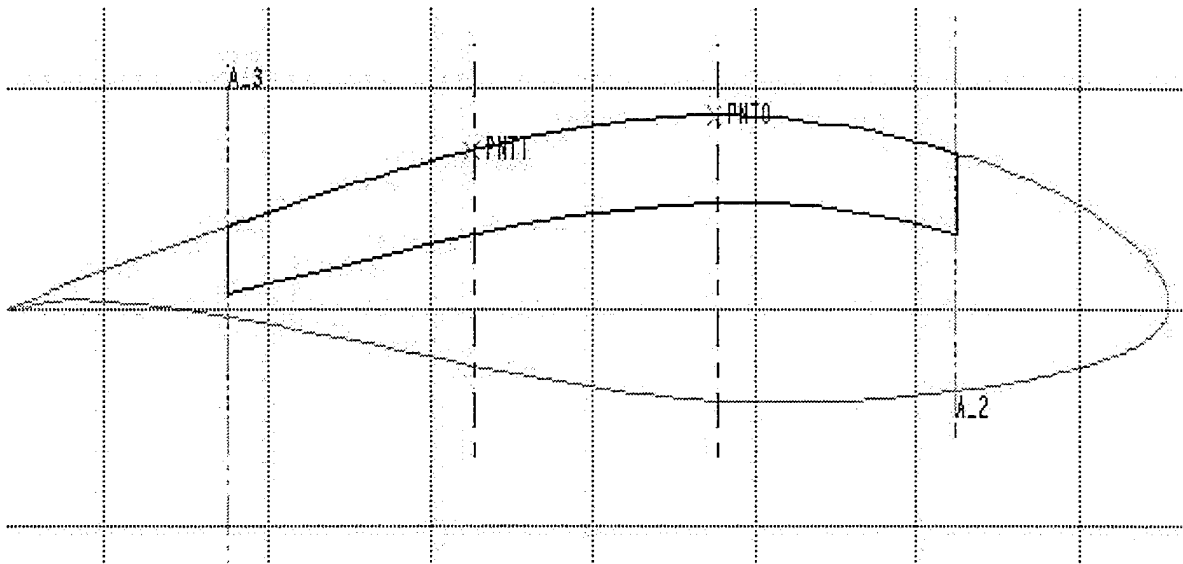
Figure(16) Completed Datum Curve sketch.

First **add two vertical centerlines**, they should go through PNT0 and PNT1, which should be located on the top edge of the brown wing section. Be sure to align the centerlines to those points.

Use the Use Edge funtion to copy the top edge of the brown section

Draw a spline, well within the wing, parallel to the top edge of the wing section, from the front axis to the back axis. The spline should have 4 points. One on the axis you start on, one on each of the centerlines you just created, and one on the other axis.

**SKETCHER(Sketch) / GEOMETRY(Adv_Geometry) /
ADV_GEOMETRY(Spline) / SPLINE_MODE(Sketch_Points) / TAN-
GENCY(Both)**

Sketch in two short lines connecting the ends of the spline to the top edge of the wing along each axis. Align these lines to the axes they are on.

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Trim the copied top edge so that it goes only as far as the axes.

Six dimensions will have to be added. Four are the distances between the four spline points and the the top edge of the wing. The two others are the angles at each end of the new spline. (If you didn't choose Both from the TANGENCY menu when creating your spline, it will be impossible to regenerate the sketch with these dimensions. If there is a problem, delete the spline and redraw it with the correct choices highlighted.)

Pick **SKETCHER(Dimension)**. To dimension the distance from the endpoints of the spline to edge of the wing, just **<LMB> on each end of the short connecting line** you drew and **<MMB> to place the dimension.**

Dimensioning the distance from the two internal spline points to the edge of the wing is slightly harder. First, **<LMB> somewhere on the spline**. The four spline points will show up. **<LMB> the spline point you are dimensioning**. The other end of the dimension will actually be at the datum point (PNT0 or PNT1). Use Query Select to pick the datum point. **<RMB> to enter the Query Select mode, <LMB> starts the Query in a certain area**, successive <RMB> clicks cycle through geometry in the area. There are several features in the area of the datum point so it may take **several <RMB> clicks to highlight the datum point in red**. Once the datum point is highlighted in red, **<MMB> to accept it as the feature to dimension to**. Now **<MMB> to place the dimension**. The dimension will now show up at this point. You will have to first choose **DIM_PNT_PNT(Vertical)** from the menu that just popped up. As soon as you do, the dimension label will appear.

You have dimensioned the angles at the end of a spline before. Just to review, it is five steps. **<LMB> on the spline, <LMB> on the spline again, <LMB> on the spline endpoint** (vertex of the angle), **<LMB> on the other side of the angle** (end axis), and **<MMB> to place the dimension.**

At this point, you should be able to successfully regenerate the sketch. If you can't, review the directions for dimensioning and aligning. Once you can regen-

erate, add two more angular dimensions between the end axes and the top edge of the wing on either side of the sketch. **<LMB> on the top edge, <LMB> on the angle vertex, <LMB> on the end axis, <MMB> to place the dimension.**
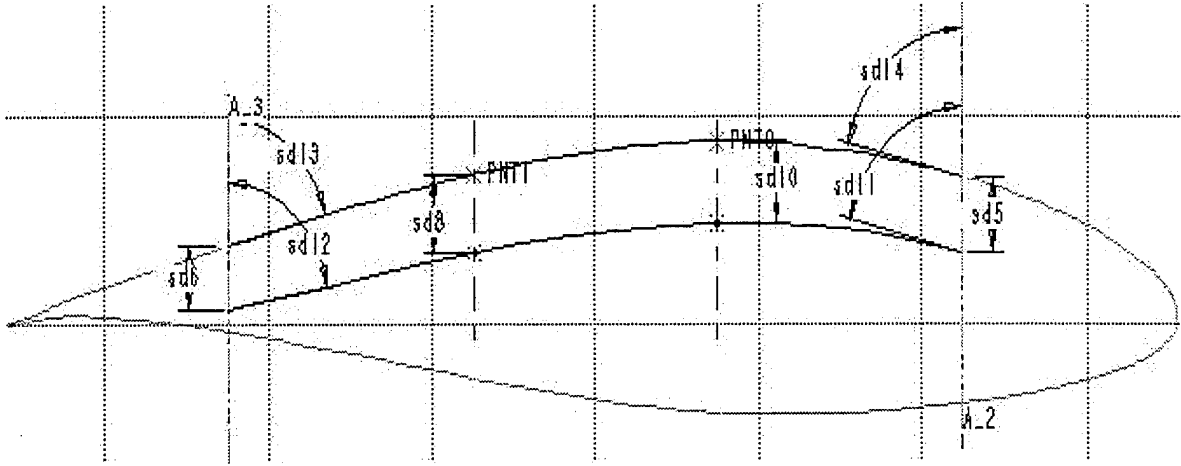
Figure(17) Dimensioned Section

Add relations making the angle between the spline and the end axis equal to the angle between the top edge of the wing and the end axis.(for example, relations of sd11=sd14 and sd12=sd13 would be added for the Figure 16 sketch)

Regenerate again. The sketch will regenerate successfully, but you will get an error message telling you that there are 'EXTRA DIMENSIONS FOUND', the two you just added will be red, showing you that they can be deleted. This one time only, **IGNORE THE ERROR MESSAGE.** Rather than define the end angles of the spline with the spline points, you are going to base them on the angles between the top edge of the wing and the end surface.

Select **Modify** and **make the all of the distances from the top edge to the spline equal to 1. Regenerate** again. Pick **Done.** You will see your new brown datum curve.

Go into layers. **Create a 'Skindtm_top' layer. Put the datum points in the Rib1 layer and the new datum curve in the Skindtm_top layer. Turn Skindtm_top off.**

Save the part.

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Turn off all layers except Rib2, Spar2, Spar3 and Surface. Starting from the creation of the datum points, repeat all steps to make the Rib2 datum curve. Be sure to do the following things.

1. Be sure that your sketching plane is the Rib that you are working on. It is very easy to pick the wrong rib and end up making your whole sketch in the wrong plane, also be sure that the direction for the sketching plane is Top/DTM2. 2. Select Both from the TANGENCY menu before drawing your spline. 3. Align when neccessary, and zoom in when there are many objects in the area of the objects you are aligning to make sure you align the correct entities. 5. Add the extra angular dimensions and relations.

Repeat the procedure for Rib3 and Rib4.

At this point, you have the datums neccessary to make the top skin solid. However, since solids cannot be hidden away in layers, the top skin will get in the way of your view when you are making the datums for the bottom skin if you create it now. Create a Skindtm_bot layer, and then repeat for Ribs 1-4 the whole precedure for creating the datum curves. This time, of course, use the bottom wing surface. You will have to use Query Select to pick the bottom wing surface when you are creating your datum points, and the sketches for the datum curves will be drawn along the bottom edge of the brown wing sections. **Put each of the new lower datum curves in the Skindtm_bot layer.**

Save the part.

TABLE 5.		Current Layer Information	
Layer Name	**Status**	**Items in Layer**	
Construction	Off	3 construction datum lines	
Trajectory	Off	3 trajectories	
Surface	Off	wing surface	
Rib1	Off	Rib1 datum plane and curve, 2 axes, 4 points	
Rib2	Off	Rib 2 datum plane and curve, 2 axes, 4 points	
Rib3	Off	Rib 3datum plane and curve, 2 axes, 4 points	

TABLE 5.	**Current Layer Information**	

Layer Name	Status	Items in Layer
Rib4	Off	Rib 4datum plane and curve, 2 axes, 4 points
Spar1	Off	Spar1 datum plane and curve
Spar2	Off	Spar2 datum plane and curve
Spar3	Off	Spar3 datum plane and curve
Spar4	Off	Spar4 datum plane and curve
Ribsurf	Off	4 rib surfaces
Sparsurf	Off	4 spar surfaces
Boxmesh	Off	wingbox datum curve mesh
Skindtm_top	On	4 top skin datum curves
Skindtm_bot	Off	4 bottom skin datum curves

When finished, turn off all layers expect DEF_DTMS and Skindtm_top. Switch to a default view and zoom in so that the four curves are easy to see, but all on the screen.
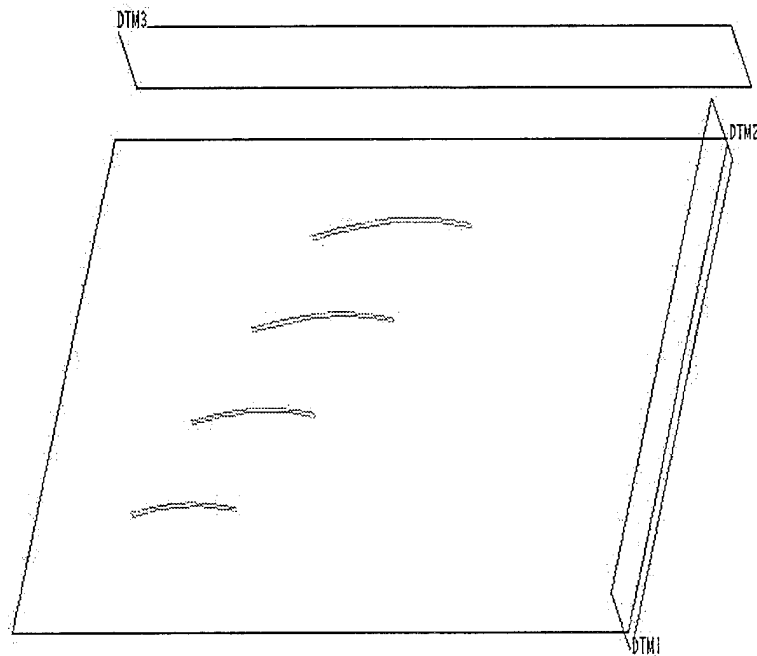
Figure(18) View configured to create the top skin.

FEAT(Create) / FEAT_CLASS(Solid) / SOLID(Protrusion) / FORM(Blend, Solid, Done) / OPTIONS(General, Open, Smooth, Regular Sec, Done) / TANGENCY_OPT(No_Opt_Tan, Select_Sec, Done)

Choose **CRV_SKETCH(Pick_Curve) / PICK_CURVE(Sel_Loop)** and **pick a point on the Rib1 curve** and then **regenerate**. A blue arrow will appear attached to one of the corners of the datum curve. **Pick Done. Pick Sel Loop again and then pick a point on the Rib2 curve**. Try to pick the same point as you did on Rib1 (ex. If you picked just to the right of the top left corner on Rib1, pick just to the right of the top left corner on rib2). **Regenerate**. You will see a blue arrow attached to one of the corners again. **The arrow must be attached to the same corner and pointing in the same direction as it is for Rib1. If it is, Pick Done. If it isn't, pick CRV_SKETCH(Start_Point) and then pick the correct corner. When it is oriented correctly, regenerate again and pick Done**. Next you will be asked 'continue to next section?' *MSG: Y*. **Pick Sel Loop and choose the Rib3 curve, again make sure that the arrow is oriented correctly and don't forget to regenerate. Repeat the process for the Rib4 curve**. After setting up Rib4, answer 'Do you want to continue the section' with *MSG: N*. The solid skin will now be created.

Save the part.

Use the same procedure to create the bottom skin.
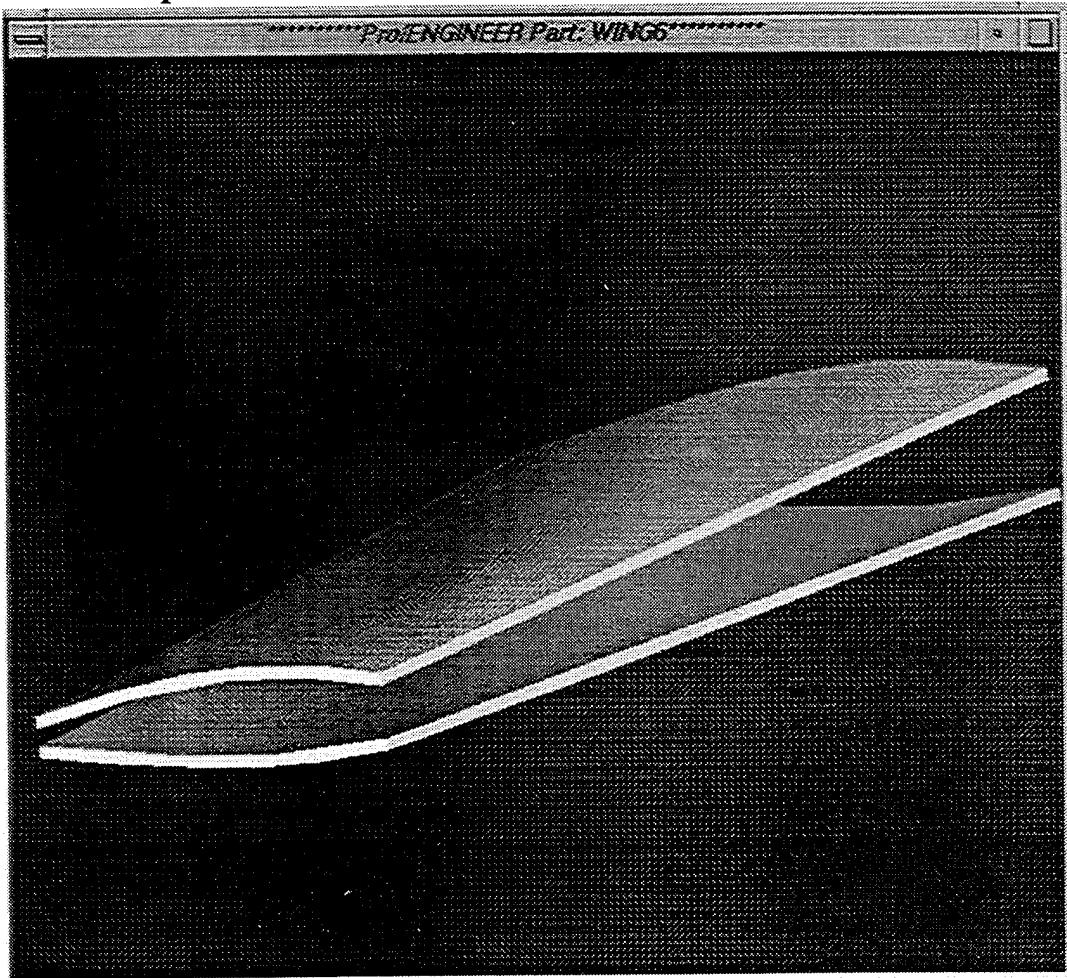
Figure(19) Shaded view of top and bottom skins

SECTION VII

Adding solid ribs and spars

Again, **open wing.prt**. Everything you need to create solid ribs and spars is already in place. **Turn off all layers except boxmesh, surface, and DEF_DTMS**. Be sure you are in a default view.

> **FEATURE(Create) / FEAT_CLASS(Solid) / SOLID(Protrusion) / FORM(Extrude) / FORM(Solid) / FORM(Done) / SIDES(Both_Sides) / SPEC_TO(Up_to_Surface) / SPEC_TO(Done)**

Now **<LMB> on the wing surface**. The top half of the surface should turn red.

> **SPEC_TO(Up_to_Surface) / SPEC_TO(Done)**

Use Query Select to pick the bottom wing surface. Pick Done until you are asked for a sketching plane?. **<LBM> DTM2**. For the direction, pick **Bottom/ DTM3**.

You will make the rib1 solid first. From the sketcher, **zoom in on the rib1 line** on the boxmesh. Draw a parallogram with two edges on the end spars and two edges parallel to rib1.
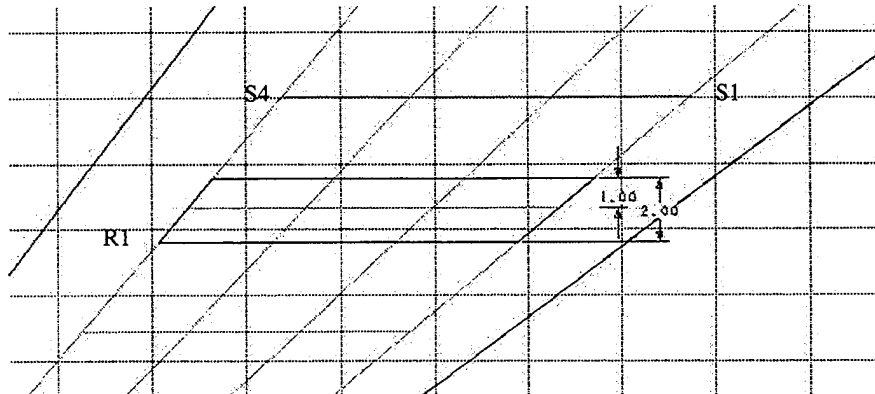
Figure(20) Completed sketch for Rib Solid including dimensions..

Two alignments and two dimensions are necessary. **Align the left and right edges to the spar lines that they are on. Dimension the distance between the horizontal sketch lines and the distance from the top horizontal sketch line to the rib1 line.**

If your top and bottom lines were not sufficiently parallel to rib1, the sketch will not regenerate successfully. The easiest way to fix this is to delete the bad lines and redraw the sketch. TRIM can also be useful in making a good parallelogram. Remember, the sketch lines can be as close to rib1 as you want as long at you can tell everything apart well enough to dimension and align properly. Dimensions can be changed later. As soon at the sketch regenerates properly, you can pick done to create the solid.

At some point you will want to **add a relation that will center the rib1 solid on the rib1 datum curve** (make the distance from the rib to the top sketch line .5 times the distance between the sketch lines). It may be easiest to do this while still in the sketcher. **Make the total thickness of the rib solid 2.**

This method can be used to create all four rib solids.

Rib solids must also be created at the root and tip of the wing. These solids will be created in the same manner, but rather than center them on Rroot and

Rtip, they should have their outer edge on these lines, so that the whole solid is contained within the wing. See figure 15 for an example.
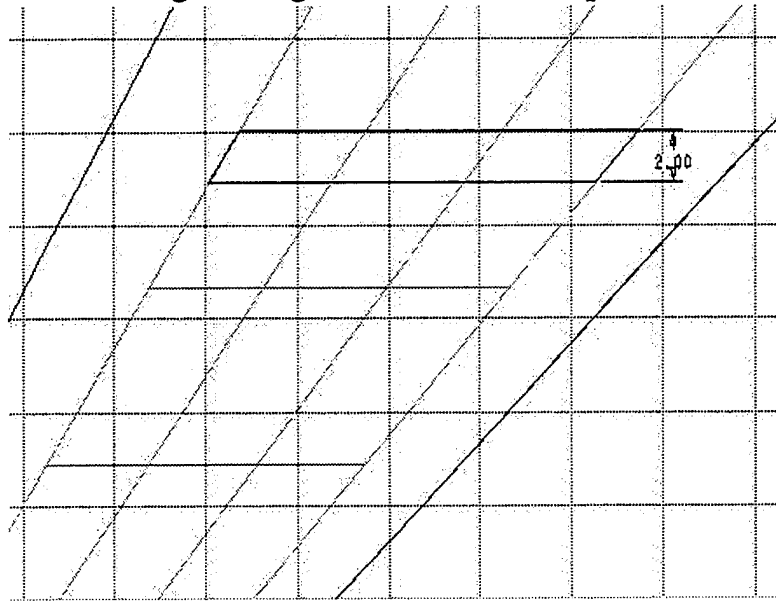

Figure(21) Root Rib solid sketch.

Construct the four Spar solids using the same method.

Add relations so that the four rib and four spar solids are centered on the boxmesh lines that they are created on. Also add relations to make all of the ribs solids and all of the spar solids the same thickness.

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Solids cannot be put into layers, so for now, they are stuck on your view. Turn off all layer and shade the resultant display to see the whole set of solids.
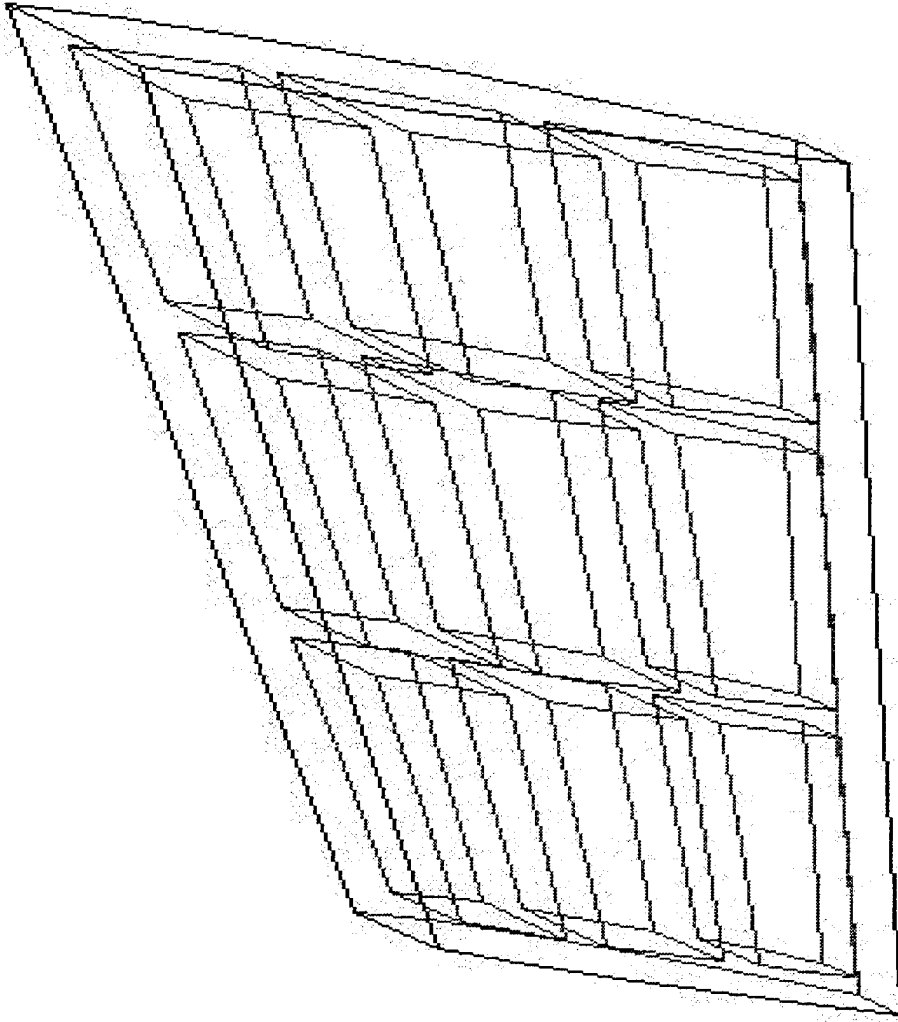
Figure(22) Completed solid ribs and spars.

SECTION VIII

Creating a Fuselage

In this section you will start a new part. Use the **DBMS(Make_PART)** command (from Section IV) **to start a new part called 'fuselage.'**

To make things clear, rename your default datum planes. **Change DTM1 to Nose, DTM2 to Waterline, and DTM3 to Symmetry**. (These names can be whatever makes sense. Wing.prt was oriented this way, but you didn't rename the datum planes.) The CSO will be the front vertex of the fuselage.

First, make several new datum planes, all offset from Nose. These will be reference positions along the length of the fuselage. It doesn't matter how many, or what value the offsets are, but put in at least five, and set the offsets and names so that you understand what is what.[1] (For example, put a Radar_box plane 80 from Nose, A Duct_inlet at 150 from Nose, a Cockpit_back at 225 from Nose, an Engine_inlet at 400 from Nose, a Tail at 475 from Nose and an Exhaust at 500 from Nose.)

Exact details and dimensions will not be given for the fuselage. If you follow the suggestions, your fuselage will most closely resemble the one pictured in the diagrams. Feel free to make a radically shaped fuselage and use modified methods to define it. This section only describes one of many methods that you can use.

Before working on the fuselage surface, you will create several basic internal components of the aircraft. These may include a box for radar equipment in the

---

1. When entering the offset value, pay attention to the green arrow on Nose. You want to offset the new planes to the in the positive x direction, so you will probably have to enter a negative offset distance.

nose, a cockpit, and an engine. (The plane will be a single engine, F-16 like, jet.) The fuselage will then be shaped around these vital features.[1] Add your dimensions in such a way that it will be easier to make changes in the future.

First, create a surface to represent the engine.

Feature / Create / Surface / SURF_FORM(Revolve, Done) / OPTIONS(360, One_Side, Capped_Ends, Done)

Pick Symmetry as the sketching plane and Top/Waterline to orient the sketch. To make a revolved surface, you will **sketch a closed cross section and an axis of revolution.**[2] The axis of revolution for the engine solid will be the intersection of Symmetry and Waterline (the lengthwise axis of the plane). Since you are sketching on Symmetry, you should just place a horizontal centerline on Waterline as your axis of rotation. Sketch and dimension an engine cross section like the one pictured in Figure 23. The front (intake) edge of the engine should, of course, be aligned to the Engine_inlet datum plane. Dimensions should be referenced from this plane when possible as well. Once the sketch is properly dimensioned and regenerated, pick **Done** to create the surface.
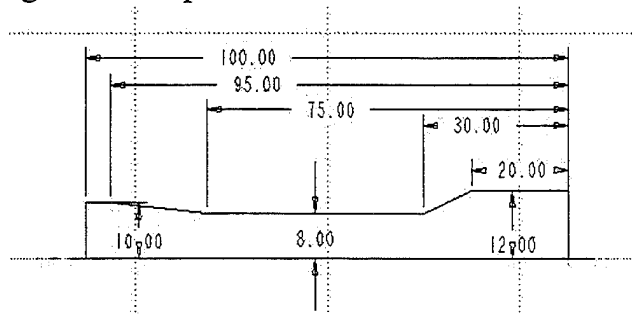
Figure(23) Completed engine sketch.

Next, create a flat, circular surface (method similar to the rib and spar surfaces), centered on the plane's lengthwise axis,[3] on the Radar_box datum

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1. If necessary, you can add more features than the three listed here. To save time, you can omit some of the three. The more features you add, the more you will learn about Pro/ENGINEER.

2. Reference Pro/ENIGINEER Part Modeling User's Guide pages 6-3 and 4-31 to 4-34.

3. Align the centerpoint of the circle to Symmetry and Waterline.

plane (make the diameter of the circle about 30). This round surface will represent the radar dish.

Extrude a cube shaped surface from the Radar plane rearward (dimension each side to about 18). This box will represent basic radar electronics. Reference this surface so that it is centered on the radar dish and moves with the radar dish. Figure 26 shows the completed radar dish and box on the right.

These surfaces are easy to make with the skills you have learned up to this time, as long as you pay attention. You should be getting a feeling for setting up sketching planes, choosing One Side or Both Sides, choosing Open or Capped ends, dimensioning sketches, etc. If you create a surface incorrectly, try to figure out by looking at it which option you chose incorrectly. Then delete the surface (or just modify it if you only need to change dimension values) and make a new one to the correct specifications.

Make a datum curve to define the cockpit. You will draw the center cockpit cross section on the Symmetry plane with Top/Waterline chosen to orient the sketch. Experiment with the sketcher, and get used to drawing arcs as well as the lines that you have done before. Add necessary dimensions to define the sketch and reference it to your datum planes. In the example, the spacing of the cockpit is referenced to Nose and Waterline. There is no reason you have to do it this way.
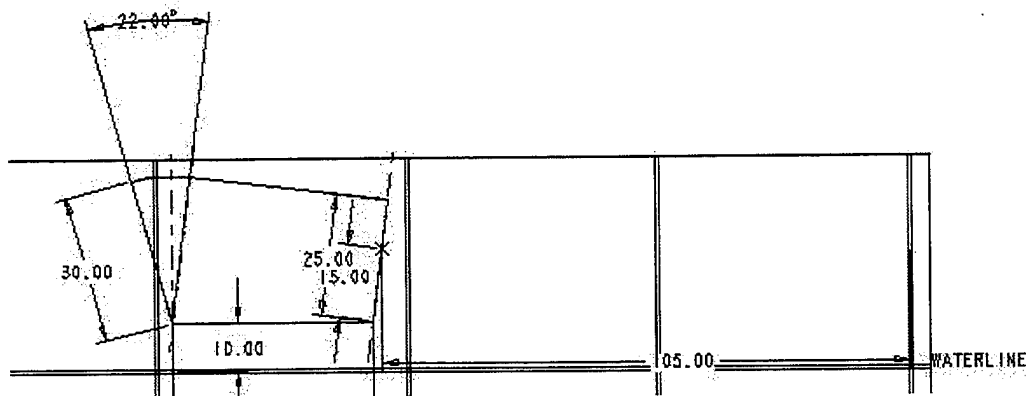
Figure(24) Completed, dimensioned cockpit sketch.

There are several methods that could be used to create a cockpit surface. Whichever method you use, you will have to take several steps. The best way to make the cockpit will depend on how you want it defined. Consider which dimensions will be base dimensions and which will be relations of those base dimensions. (Whichever method you use, you will have to sketch a second cockpit cross section, the front view. One method is to sketch your second cockpit section on a datum plane (pick Make_Datum when asked for a sketching plane) perpendicular to Symmetry and through the back line of the original cockpit sketch (the line with the dimension of 30 in Fig. 24). Sketch an upside-down, closed dome shape section and revolve the surface on an axis that goes through the bottom of the 'dome' (the vertex of the 22 degree angle in Fig. 24). Add a relation so that the surface revolves exactly as far as the curve on the original cockpit sketch (the 22 degree angle in Fig. 24). Then extrude the other end of this new surface to the front of the cockpit.[1] This new surface will be too big, it will go below the bottom line of the original cockpit sketch. Create another new surface by extruding the bottom line of the original sketch in both directions for a distance of 50.[2] Then merge[3] this surface with the one that is too big, you should end up with a closed cockpit surface.)
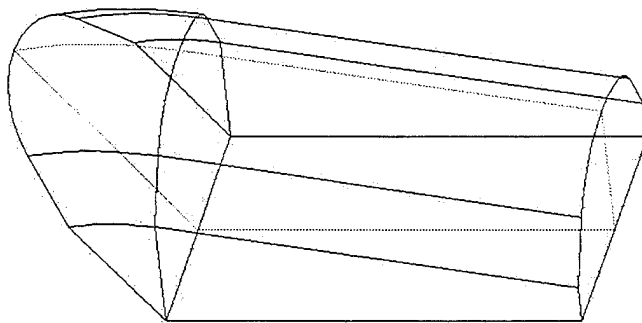

Figure(25) Isometric view of a typical, completed cockpit surface.

1. Reference Pro/ENGINEER Part Modeling User's Guide page 6-27.
2. This distance doesn't really matter, it just has to be wider than your second cockpit sketch.
3. Reference Pro/ENGINEER Part Modeling User's Guide pages 6-2 and 4-26 to 4-31.

When you are finished building the cockpit, try modifying different dimensions, such as those on the original cockpit sketch, to make sure the cockpit surface completely rebuilds itself according to the new dimensions. If you make a change that cannot be regenerated, Pro/ENGINEER will give you new menus. Options on these menus will allow you to restore your part to the way it was before you made the change or change the way the part is defined.[1] You are now finished your internal components.
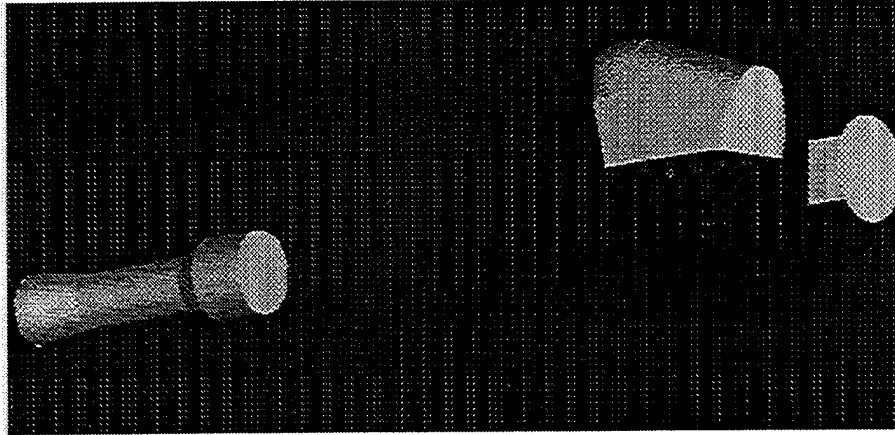
Figure(26) Three internal features, shaded view.

Save the part.

The fuselage surface will be defined by sets of curves in two directions. The first set will be three faired curves. These will be the top, bottom and side boundaries for the fuselage. The second set will be cross sectional curves on the datum planes along the length of the fuselage.

The faired curves will be created in two main steps. First, a chain of segmented straight lines will roughly define the boundary. Then conic lines will be added to smooth the straight lines.

Create a new datum curve. This curve will be the segmented line that will eventually be faired to create the top edge of the fuselage. **Pick Symmetry as the sketching plane and Top/Waterline to orient the sketch.** It is from this

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1. Using these menus is somewhat complex. Reference Pro/ENGINEER Part Modeling User's Guide Section 5.

view that you will sketch the top and bottom edges of the fuselage. **Use approximately seven segments to draw the top edge of the fuselage. The front point should be on the coordinate system origin, and aligned to the Nose datum plane. The 'highpoint' should be a horizontal line crossing just above the top of the cockpit. One point should be placed near the front vertex of the cockpit.** The rear endpoint can be aligned to Exhaust or offset from it. Exaggerate the angles in your drawing, or Pro/ENGINEER might make assumptions (such as making a line parallel to Waterline) that you don't want it to.

Dimension your segmented line so that the following modifications are possible:

1.) The angle of the fuselage coming into the cockpit window can be adjusted. (8 degrees in Figure 27.)

2.) The distance from the top of the cockpit to the top of the fuselage can be adjusted. To do this, place a vertical centerline through the bottom left point on the original cockpit sketch. Next, place a sketch point on the top of the arc of the original cockpit sketch where the centerline crosses. Align the point to the arc. Also, center your horizontal sketch line on this point. The dimension can be placed between this point and the horizontal segment. After the sketch has been successfully regenerated, modify this dimension to 0.

3.) The distance from the point near the front of the cockpit to the cockpit can be adjusted. This point will eventually be coincident with the front of the cockpit, so horizontal and vertical dimensions will have to be added between these points. Again, modify these dimensions to 0 after you have regenerated.
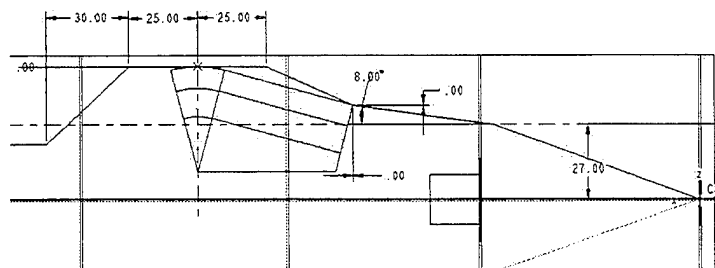
Figure(27) Front half of dimensioned, straight line sketch.

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Once the top edge is finished, you will smooth this chain of straight lines to make the top profile of your fuselage. First, you will have to add datum points at the ends of the segmented line as well as at the midpoint of each segment except the first and last.

Create / Datum / Point / DATUM_POINT(On_Curve) / PNT_DIM_MODE(Length_Ratio)[1]

This function will allow you to place a point at a certain position on a segment. The number you enter will determine the ratio of the position of your point to the whole segment. **<LBM> one of the straight sections on the existing curve** (Don't pick the end segments, you only need these points on the middle segments). Pick **Done Sel.** Enter *MSG: .5* to place the point at the midpoint of the selected segment. You will see a green point show up on the center of the segment. **Repeat the procedure so that you have midpoints on all the internal segments. Pick Done.**[2] Now, using the **DATUM_POINT(On_Vertex)** option, put points at the very ends of the datum curves.[3]

These will be your reference points for the conics[4] which will make the top profile for your fuselage.

Create another datum curve, again choosing Symmetry as the sketching plane and Top/Waterline for the direction. Zoom in on the nose of the plane. **Put a sketch point on each of the first two segments.** Put the one on the first segment about in the middle (Sk1 on Figure 26) and the second one to the right

1. Reference Pro/ENGINEER Part Modeling User's Guide page 3-13.
2. You can make several datum points as one feature, or you can create them all as individual features. Each time you pick Done, the current feature is finished. If you make eight points without picking done, they will all be part of the same feature. How you decide to group your datum points into features depends on how you plan to setup your layers.
3. Reference Pro/ENGINEER Part Modeling User's Guide page 3-13.
4. Reference Pro/ENGINEER Part Modeling User's Guide page 2-12.

of the midpoint datum point you placed earlier (Sk2 on Figure 26). **Align each of these points to the segment they lie on.**
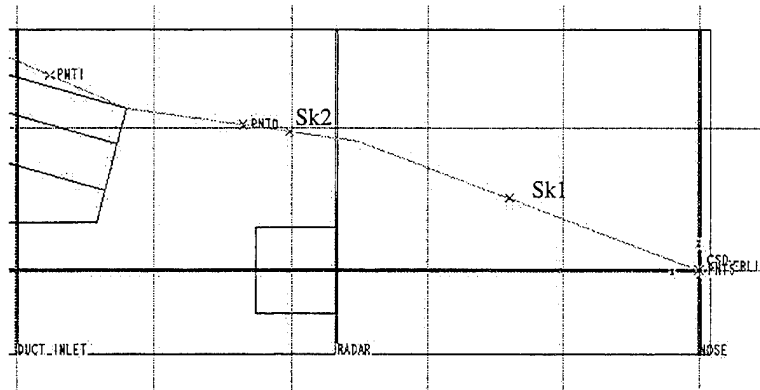
Figure(28) Sketching view for first curved datum with sketch points added.

Now add a centerline through each of these two sketch points, perpendicular to the line it is on.

GEOMETRY(Line) / LINE_TYPE(Centerline, Perpendicular)

<LMB> on the line segment, then <LMB> on the point. Pick Perpendicular again and create the second centerline. Next, connect the two sketch points with a conic.

GEOMETRY(Adv_Geometry) / ADV_GEOMETRY(Conic)

<LMB> on one sketch point, then <LMB> on the other sketch point. Move your mouse pointer around to see the red, paraboloid-like conic. Place your mouse pointer so that the conic creates a smooth curve between the two segments that the sketch points are on. **<LMB> to finish the conic.**

Five dimensions must now be added to your sketch.

1.) The distance from the point on the front vertex of the plane to the centerline that goes through the sketch point on the first segment.

2.) The distance from the other centerline to the middle datum point on the second segment should be added. These dimensions are placed using the standard method, pick the objects to dimension to with the <LMB> and place the dimension with the <MMB>.

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3.) Dimension the conic itself. <LMB> on the conic, and <MMB> to place the dimension.

4,5.) Dimension the angles between the conic and the perpendicular centerline at each end of the conic. <LMB> the conic, <LMB> the conic endpoint, <LMB> the centerline, and <MMB> to place the angular dimension. Regenerate to make sure everything is in place.
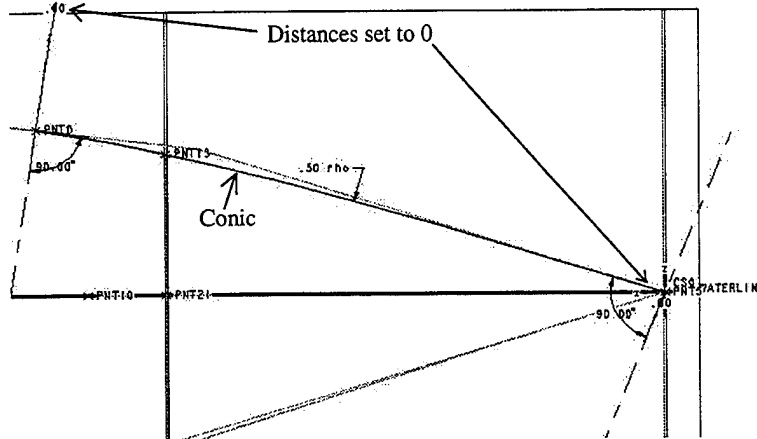
Figure(29) Dimensioned, regenerated conic sketch, values edited[1]

Modify the dimensions so that both distances are 0 and both angles are 90 degrees. Making the distances 0 will cause all of your curved segments to meet. Making the angles 90 will make the curves tangent to the original straight lines. The conic dimension will be automatically set to .50. **Regenerate again and pick Done.** You will see a brown datum curve from the front vertex of the plane to the midpoint of the second segment which 'smooths' the first corner of the segmented line.

Repeat this process to smooth each corner on the segmented line. For the internal curve sections, place your sketch points between the datum points on the two segments and the corner you are smoothing. Place the distance dimensions between the datum points and the centerlines. **Remember to:**

1.) Align the sketch points to the lines they are on.

2.) Modify your distances to 0 and your angles to 90 degrees.

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1. This means the distances have been set to 0, so the dimensions are hard to see.

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3.) Dimension the conic itself.

For the last curve, extend the conic all the way to the end of your original segmented datum curve, as you did at the nose.

You now have several individual conics that make up the first faired curve. You will need one continuous curve when creating the fuselage surface. To make this continuous line, **create another datum curve, again sketching on the Symmetry plane. Use the Use Edge function to copy all of the curved lines. Pick Regenerate and Done**. Now you have a datum curve that is a combination of all the short curves.

Save the part.

The curve for the top edge is finished. Now **repeat the whole procedure, starting from the creation of the segmented datum curve, for the bottom edge and the side edge**. To save time, you may want to make the top and side edges out of only four or five line segments. **Remember to align the ends of the segmented lines. Also remember to add your dimensions so that the eventual surface will be shaped around your internal features**. If the size or placing of those features changes, the top, bottom, and side curves should move to stay out of the way.

When constructing the side edge, your sketching plane will be Waterline. Also note that you only need to make one side sketch, as only half the fuselage needs to be built since it is symmetric.

If you haven't already, this would probably be a good time to setup some layers. You will want to remove the original, segmented lines from your display, as well as the datum points that marked the midpoints of those segments. If you want to create a layer (or layers) for the internal surface structures, you can do that as well.

The datum curves that will define the length of the fuselage have been completed. The cross-section curves must be completed as well. One of these curves

will be constructed on each of the datum planes along the fuselage (all of the planes offset from Nose). Since both the faired curves and the cross sections will be used to define the same surface, they must cross at certain points. These points are on the datum planes that the cross-sections will be sketched on. To construct these points, use the following commands:

Create / Datum / Point / DATUM_POINT(Curve_X_Srf)[1]

Next **<LMB> one of your three faired curves**, use Query Select to make sure that you pick the whole curve and not just one of the pieces. **Next you will pick datums that the line intersects.** If you want, you can pick all of the datum planes that cross the fuselage right now, or you can just pick the first one.[2](You probably have points at the endpoints of the faired lines, so you won't need to do this for the Exhaust plane.)

You will see a numbered, labeled datum point show up. **Repeat the process for the other two faired curves.** Take careful note of (or better yet, write down) the numbers of the three points on each datum plane. You will be sketching a curve on each plane and will need to know which points to align to for each sketch. (The best solution, of course, is to create a layer for each of the datum planes offset from Nose. Into each one put the plane itself, the three points on it, and eventually the cross-section curve you sketch with those three points. Then turn off all of these layers except the one you are working on.)

To draw the first cross-section curve, again **create a datum curve**. The sketching plane will be the first plane to cut the fuselage from the nose (Radar_box), and the orientation will be Left/Symmetry. Unless you separated your points into layers, this view will show many points, possibly very close together (see Figure 28). **Draw one conic from the point on the top curve to the point on the side curve and another from the point on the side curve to the point on the bottom curve.** MAKE SURE YOU PICK THE POINTS THAT ARE ACTUALLY

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1. Reference Pro/ENGINEER Part Modeling User's Guide page 3-12.

2. Again, this is a matter of how you want the points grouped into features. Your plans for constructing layers determine how you want to construct your datum points.

ON YOUR SKETCHING PLANE. (By spinning the view you may be able to see which points are on your current sketching plane, but remember to pick Sketch after your spin to return to the sketching view.) **Align the endpoints of your two conics to the points they start and end on.** <u>You will almost always need to use Query Select to be sure that you pick the right points</u>. **Dimension the angle at each end of each conic and dimension each conic itself.** Set all four angles to 90 degrees.
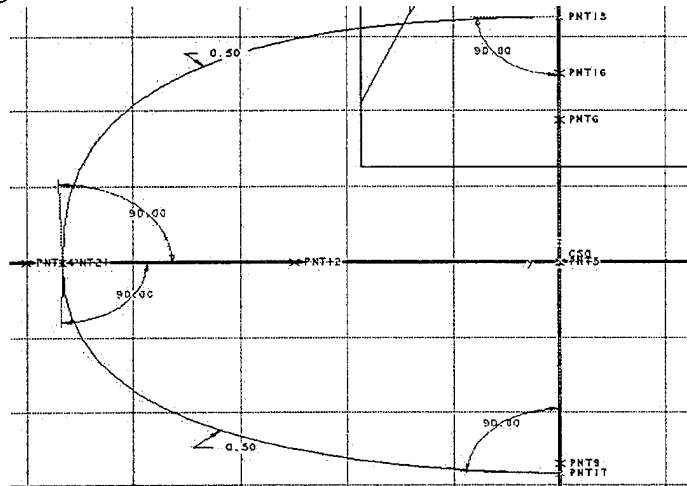
Figure(30) Completed, dimensioned cross section curve.

Repeat the procedure for each of the other planes that cross the fuselage, including Exhaust. You didn't need to add the three datum points to the Exhaust plane if you aligned your length lines to it, as there were already points there. Be sure that you choose the correct sketching plane. It is easy to accidentally pick Use Prev when asked to setup the sketching plane and then draw your whole sketch on the wrong plane.

You should now have everything in place to create your fuselage surface. Check a few things to make sure the surface will be constructed properly. Be sure ALL the distances on the sides of your conics are set to 0. In other words, make sure that your faired curves don't have any gaps. Be sure all of the cross-section curves are on the proper DTM plane. Shift to a default view and zoom in on each cross-section curve to make sure it passes through the three points on the faired curves. Make sure all three curves converge on the nose point. **Save the part.**

Now it is time to construct the fuselage surface.

> **Create / Surface / New / Advanced / ADV_SRF_OPT(Boundaries) /**
> **OPTIONS(Blended_Surf, Optional_Tan, Done)[1]**

Now you must specify curves in 2 directions to define the fuselage surface. The curves for the first direction are the faired curves.

> **FIRST_DIR(Add_Item) / SELECT_ITEM(Curve)**

<LMB> the top curve first, then the side curve, and finally the bottom curve. They should turn dark blue when you pick them. Use Query Select to make sure that you are picking the complete faired curve and not one of the sections. **When all three curves are highlighted, pick Done Dir** (Done Direction).

> **SECOND_DIR(Add_Item) / SELECT_ITEM(Curve)**

Now <LMB> the cross-section curves. Be sure to pick them in order from front to back. When they are all highlighted, again pick Done Dir.

Next you will be asked if you want to specify any control points in the first direction, *MSG: N*. Then you will be asked if you want to specify any control points in the second direction, *MSG: N*.

Finally, you will select the optional tangency. Basically, this means that you want to define the surface as normal to the Symmetry plane where the surface boundaries intersect that plane. **At this point, the boundaries of your soon-to-be surface should be highlighted in light blue. <LMB> somewhere on this highlighted line. Then pick BNDRY_COND(Norm_to_Sket, Done_Boundary). Pick TAN_COND(Done).** Assuming everything went according to plan, you now have your surface. To check that it was created properly and to make sure that your internal features are all within the surface, **turn on all layers and then shade and spin the part. Save the part.**

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1. Reference Pro/ENGINEER Part Modeling User's Guide pages 6-5 to 6-13.
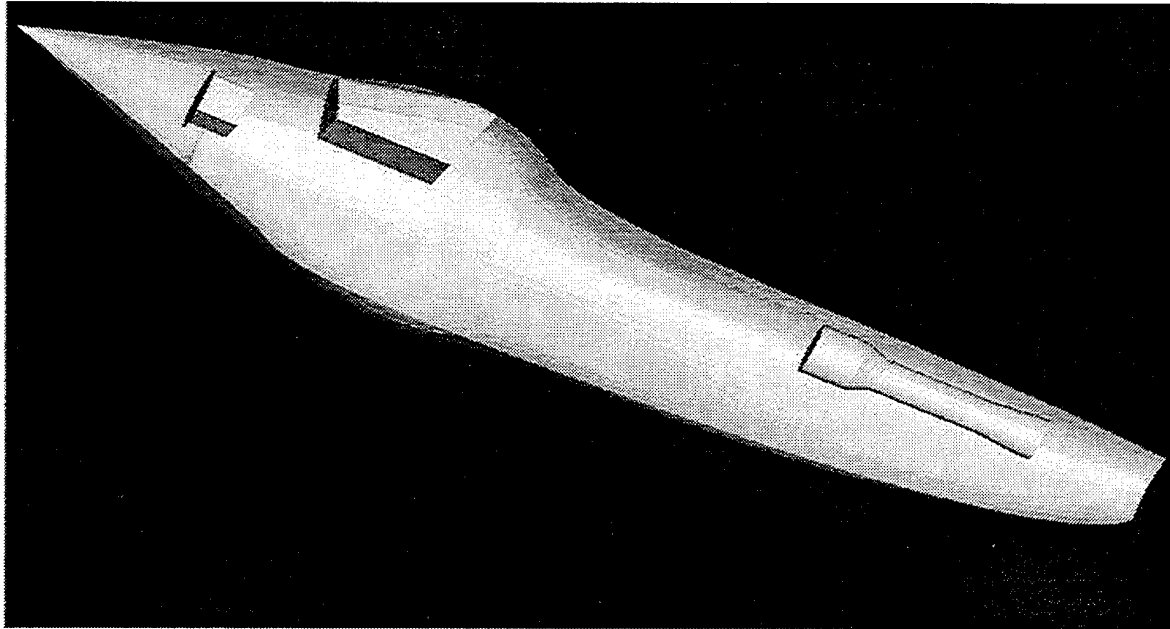
Figure(31) Shaded fuselage surface with internal features showing.

Try changing the size or location of some of your internal structures, or changing the dimensions on the sketches for the segmented top, bottom and side curves. If everything was done correctly, each regeneration should yield a new, differently shaped fuselage to fit your new dimensions.

Save and exit.

SECTION IX

References

Pro/ENGINEER Part Modeling User's Guide, Version 14, Parametric Technology Corporation, 128 Technology Drive, Waltham, MA 02154 (617) 398-5000

Pro/ENGINEER Fundamentals, Version 14, Parametric Technology Corporation, 128 Technology Drive, Waltham, MA 02154 (617) 398-5000

APPENDIX A

Using Configuration Files

The config.pro file sets the default settings for your Pro/ENGINEER system. A global config.pro file, located in the 'loadpoint' in the text directory where your Pro/ENGINEER files reside, is the first one loaded as Pro/ENGINEER starts up. After that, a more personalized config.pro, located in your home directory, is loaded. In this file you can make changes that will override the global configuration. You can also put a config.pro into specific directories where you will be running Pro/ENGINEER. This will allow you to make additional specific changes to the default setup for files you will be working on in that directory. Configuration files can be edited with any text editor.

Changes to your Pro/ENGINEER system can be made by editing the config.pro file. Here you can set macros, or mapkeys, to carry out common menu picks.[1] Many other customizations, such as automatically saving your trail files to a specific directory, can be created with the config.pro file.

Setting up mapkeys for common commands, such as view manipulation, can save you a lot of time. To set a mapkey, use the following command format:

mapkey keystroke #Menu_Choice;#Menu_Choice;...

Precede function keys by a $ sign. For example, to make a macro that shades the display each time the F1 key is punched, add the following line:

mapkey $f1 #view;#cosmetic;#shade;#display;#done-return

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1. Reference Pro.ENGINEER Fundamentals pages 10-7 to 10-10.

It is recommended to add mapkeys for shade, zoom in, zoom out, zoom reset, spin, default, repaint and save.

Buttons can be added to menus by editing the menu_def.pro file,[1] located in the same directory as config.pro. To add a menu button, use the following command format:

@setbutton MENU Button_Name "#Menu_Choice;#Menu_Choice;..." "Message for message window"

The following lines must be added to your menu_def.pro to configure the menus so that they match the ones in the manual.

@setbutton DBMS Make_PART "#mode;#part;#retrieve;default;#dbms;#save as; ;#dbms; #erase;#confirm;#part;#search/retr;" "Make part from the Default"

@setbutton SKETCHER Trim #Geom Tools;#trim;

@setbutton LAYERS Repaint "#Done/Return;#View;#Repaint;#Pan/Zoom;" "Repaint recently changed layer display"

Appendix D of the Pro/ENGINEER Fundamentals manual gives a full list and description of all the commands that can be used in the configuration files.

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1. Reference Pro/ENGINEER Fundamentals pages 10-11 to 10-15.

APPENDIX B

Graphics Creation

All graphics appearing in this document were originally taken as screen shots of Pro/Engineer on a Silicon Graphics Indigo system with Elan graphics. The Snapshot screen capture utility was used to take the screen shots. The snapshots were saved as .rgb files. These files were then converted into greyscale jpeg (.jpg) files using the utility XView. The .jpg files can be opened by Corel PHOTO-PAINT 5.0 (PC/Windows) and then saved as .tif files. The .tif files can be imported directly into FrameMaker (place an anchored frame where you want a graphic, highlight the frame, and then import the file). Text or additional lines can be added to the graphics with the FrameMaker toolbox. The graphics files were moved from the SGI system to the PC and from the PC to the Xwindows system where FrameMaker was running with Wftp in binary file transfer mode.

To save time, XView can be used to convert the .rgb screen shots into .ps (Post-Script) files. These can be brought directly into FrameMaker. The postscript files, however, are 10 to 20 times larger than an equivalent .tif file. The larger files take up more memory and slow down the FrameMaker file once they are imported. Smaller .tif files will be displayed as they look on the screen in Frame-Maker. Larger .tif files will only appear as a grey box. The huge .ps graphics files will always appear as a grey box.

Note also that because all graphics start out as screen shots, the size and resolution of your monitor affects the size of the file. A large, high-resolution monitor will cause you to end up with larger files than a smaller, lower resolution monitor.