After data collection using the Microscribe digitizer through Rhino and importing into Pro/E:

Note: This data was checked for approximate size before any thing else was done!

Datum planes are created at station locations defined by the upper datum point on the profile curve. These are not equally spaced locations, but locations defined by where you decided that cross curves would best be located.

Note the re-named layers and datums

Hint:

Insert a simple extruded feature near the top of the model tree to keep Pro/E from zooming out when working of the first Style feature. This feature can be suppressed when not needed.
Creating initial Style curves

Center Line curve – Planer
    Use the points to create a smooth curve which approximates the center line cross section.

Main Profile curve – Planer (offset at 2”). Make sure this curve ENDS at Station 1 and 9.
    To do this, use Analysis – Measure and find where these planes are located relative to the
    profile plane.
    Make sure the end point X value matches this dimensional value.

Add planer curves at the front and back defining these surfaces. Use the same methodology,
except for the end points, make one end tangent to the existing curve and define the other end to
be tangent (normal) to the center line datum plane. You know the x,y coordinates of the center
line curve. Make sure that your x coordinate matches exactly! Z will of course be 0 (on the
center line). Triple check to make sure the x coordinates match up to all displayed decimal
places. Do not trust you “eye”.
Front edge example:

The main profile curve just created must be put on a “smooth, fair” surface. The only way to accomplish this inside of a singular style feature (for updating purposes) is as follows (you can Edit – Copy a curve in style, but it will be unlinked – not good for updating purposes):

Create a 2-D surface within Style (we just did that)
Drop the planer curve onto the style surface.

There are issues when doing this. First, how to create the 2-D surface since there is no copy or offset planer curve functionality in Style (and still be linked).

To generate a 2-D surface do the following.
1) Produce a planer curve (on the center line) defining the bottom edge of the remote.

Notice that the curve simply uses the points as rough reference locations. The endpoints must to located at the ends of the centerline curve.

Addendum 2/17/05
Extend the planer curve so that it extends beyond both ends of the center line curve. This is so we are sure that there is always a 2-D surface in which to drop the upper (main profile) curve onto. Even though tangencies are defined properly for the upper curve and it initially drops onto the surface, later redefinitions of the Style feature will generate a failed feature upon exiting the Style feature due to the fact that there is now no short section of the dropped curve...
near the centerline. This in spite of the fact that a Regenerate All does not indicate an error. See the following figure to understand how the geometry should extend farther out:

2) Create an offset plane (Styling – Internal Plane)
3) Create four style curves on this plane defining a large rectangular surface.

4) Create a Style surface using these four curves
5) Drop the 2-D curve created in step 1 on this surface.

6) Create a new style surface using the two curves.
You now have a surface that we can use to project the 2-D representation of the main profile on, after we add the 2-D representation of the front and back edges.

Eventually, when we are satisfied with our part geometry, we can delete the large vertical surface. Select the surface, Edit – Delete and Unlink the 2-D curve. Regenerate and select “Convert selected curve entities”. This will leave the second curve that was projected onto the vertical surface hanging in space. It is no longer parametric, so don’t do this until the model is essentially done.

Drop these curves on the 2-D surface.

Trim the surface using the Trim Selected Quilts tool. Make sure you follow the prompts and select the portion you wish to delete.
Showing most curves and extra surfaces hidden:

Hint:

It might make sense to change the color and transparency setting for the vertical surface as hidden attributes set in Style do not translate back to the part level upon leaving the Style feature. This has to be done outside of the Style feature.

Create curve at Station 5

End points are only snapped to the center line and main profile curve. Note that only the points (light green) for Station 5 are turned on.
Do the same thing to generate curves at Station 1 and 9.

A quick test surface is produced to make sure curve end points actually intersect as expected. The curve at station 5 is used as an internal curve.

Test to check the ability to add end surface:
Note the fact that the surface does not shade, yet the continuity conditions are still displayed in the figure on the right. The system is trying to tell you something. We will see what this issue is a few pages later.
What makes G1 continuity possible in both directions?
   The fact that the bottom curve at the front edge is tangent (at its endpoint) to the main profile curve and tangent (perpendicular) to the frontal (centerline) datum plane.

Can this surface be created by selecting on the center line curve, the main profile curve, the curve at Station 5 and the curve at the bottom front edge with the curve at Station 9 used as an intermediate curve?
   No. Surfaces cannot be created using boundary curves that are tangent to each other (main profile curve and bottom edge curve).

After creating some temporary surfaces at the other end, exiting Style, saving the part, and only then:
   A) Removing (temporarily) the vertical, offset Style surface. Refer to page 5 on how to effectively do this.
   B) Creating a large surface at the centerline
   C) Merging the Style and just created large surface at the centerline
   D) Solidifying the geometry
   E) Mirroring the entire part......
Note: There is part of a line missing at the front. Pro/E indicates a geometry check at this location. The Style surface in this location does <b>not</b> shade (it will mesh if wire frame mode is active) at the Style level, indicating that there are issues using these curves. In addition, during the merge operation, a sliver of surface material was “left over” when viewing results of keeping select sides which is also a hint there is a problem with the surfaces. Additional modifications will need to be done to produce this surface properly.

Modifications from creating the temporary surfaces on were <b>NOT</b> saved as this is a quick and dirty check to make sure things are going to go together as expected. Waiting until all the curves and surfaces are produces before checking sometimes leads to unexpected results (and a lot of time and effort down the drain).

The missing line geometry happens if you are lucky. If is just as possible that Pro/E will exit when you try to Mirror the geometry. If that is the case, it is possible to try to produce a solid part by Mirroring the Style feature and doing a Merge – Join. You still have to remove the vertical offset Style surface to be able to merge the original and mirror feature. The net result is you can see that this will not Solidify properly due to the one sided edges at the front center of the part (small triangles).

How to resolve the issues at the front? Three sided surfaces can be somewhat problematic at times. The following figure is a result of creating a surface using three curves. Note the fact that the surface is not shaded, but continuity conditions are displayed.
Cancel this surface definition and add a free curve as shown below.

Generate a four sided surface.

Drag the curve down toward the corner and create another surface to fill in the corner (using three curves). Note the tangency at the one corner (Normal to the center datum). Pay attention to the connection conditions.

You might have to click on the continuity indicators to increase the connection at the other end to G1.
Once you have all your tangencies and continuities set correctly, the following will be the result of the “quick test”. The right most view is shown with no display of tangent edges (Tool – Options).

Do **not** continue past this point if you cannot do this!

Showing all surface connections:

Reflection analysis shows:
Of course this doesn’t properly represent our model. The top edges are too sharp and the curvature near the front is incorrect. We need additional curves to help drive the surfaces.

After adding internal curves at stations 3, 6, 7 and 8 and a quick slope check:

It appears that station 5 needs to be made a little rounder and station 9 and 10 could be a little sharper to correspond with the clay model.

After changes (they are subtle, but they end up being important):
Unfortunately, this causes a wrinkle in the “nose”, which has to be dealt with.

It was decided that the best way to approach this problem would be to remove the surfaces (large and tiny) that defined the front, add in an additional 2-D curve offset from the horizontal, and rebuild the surfaces.

Of course the tangent connection was set to Surface Curvature for the left side of the new curve and this allows us to generate the following surface connections:
Reflection at nose.

We have still maintained our tangency conditions as can be seen by the shaded and no hidden representation below.

It is now time to create a .stl file from this data and print it on the Z-402 printer to see how it compares with the clay model.