A CAD Approach to Integrating NDE With Finite Element

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Summary

Nondestructive evaluation (NDE) is one of several technologies applied at NASA Glenn Research Center to determine atypical deformities, cracks, and other anomalies experienced by structural components. NDE consists of applying high-quality imaging techniques (such as x-ray imaging and computed tomography (CT)) to discover hidden manufactured flaws in a structure. Efforts are in progress to integrate NDE with the finite element (FE) computational method to perform detailed structural analysis of a given component. This report presents the core outlines for an in-house technical procedure that incorporates this combined NDE–FE interrelation. An example is presented to demonstrate the applicability of this analytical procedure. FE analysis of a test specimen is performed, and the resulting von Mises stresses and the stress concentrations near the anomalies are observed, which indicates the fidelity of the procedure. Additional information elaborating on the steps needed to perform such an analysis is clearly presented in the form of mini step-by-step guidelines.

Introduction

Nondestructive evaluation (NDE) is a critical technology that is very crucial for the improvement of the design and the quality of a product in a cost-conscious production environment. NDE detects variation in the behavior of a material or component to find hidden defects and anomalies without altering or damaging the test piece. Using such available techniques to update the production process requires characterization of the faults and their influence on the component, warranting extra work. For instance, supplementary investigation via an experimental study, computer simulation (e.g., the finite element (FE) method), or a series of control tests may be performed. This task is dependent on the material being tested

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and on the structural complexity of the component, two parameters that have increasing importance in modern manufacturing.

Furthermore, imaging or digitizing an object is a typical approach used to generate a computerized three-dimensional reconstruction to be integrated into a particular design. This is generally referred to as rapid prototyping and is the ability to build high-quality physical prototypes directly from computer-aided design (CAD) files. Using rapid prototyping, full-scale models or patterns can be built using a variety of materials in a fraction of the time required by more traditional prototyping techniques (refs. 1 and 2). For instance, Velocity^2 (ref. 3) is a common rapid prototyping software that is used to carry out the construction process of three-dimensional volume models and the subsequent generation of a stereolithography (STL) file that is suitable for CAD applications. Production of three-dimensional models of objects from computed tomography (CT) scans is becoming a key to NDE methodology (refs. 4 and 5).

The objective of this work is to develop an NDE–FE link using the image processing software Velocity^2, a CAD-based software Rhinoceros 3D (Rhino3D) (ref. 6), and the MSC/Patran (ref. 7) finite element graphical processing package to enable the construction of a three-dimensional model using a series of CT scan images. The goal is to generate a meaningful finite element model ready for analysis and able to model all the structural anomalies detected via the NDE process as closely as possible to their original shape.

Appendix A presents a table that lists and describes the Rhino3D toolbar commands relevant to creating the solid to be processed with MSC/Patran. Appendix B gives a brief description of an in-house program translator used to convert STL files to "Patran neutral files" (PNF).

Analytical Approach and Modeling

This section illustrates the analytical operations carried out in the development of the methodology applied to attain an image-rendering scheme to construct three-dimensional models of an object. Also, a finite element model for analysis is constructed.

Three-Dimensional Volume Rendering

The objective of this procedure is to model regions of lower density within a material (cracks, voids) so they can be analyzed via finite element analysis. To create these models, the image processing software Velocity^2 is used to construct a rudimentary three-dimensional model from a series of two-dimensional CT scans. Figure 1(a) depicts a sample CT scan used in this report. The specimen imaged is a rectangular section of a ceramic composite—a material typically containing voids and porosity identifiable with NDE methods. The procedure outlined and described in this report proceeds using this image and the data obtained from it.

Image enhancement.—Before the three-dimensional geometry is created, image enhancement is often required to clean up the CT images. For example, images need to be enhanced to make certain Regions of Interest (ROI), such as the edges of cracks and voids, better defined. Image enhancement is done by using tools such as filters and grayscale operations. Velocity^2 has a variety of tools for image enhancement (Logarithm, Median, Scale) that can be applied using user-defined scripts to a series of images.

IGES file from Velocity^2.—The Initial Graphics Exchange Specification (IGES) is a widely supported CAD file capable of both two- and three-dimensional data types. Rhino3D requires that an imported file be in a CAD type format. Velocity^2 permits export in the IGES file format that is also supported by Rhino3D. For an IGES export to work, Velocity^2 requires that an ROI mask be placed on the images to show the boundaries of the IGES curves. This is done by using Velocity^2's Logic Operation

Flood Fill All and setting the proper threshold value to highlight the desired regions. After image enhancement and IGES file generation, the slice in figure 1(a) is converted to curves outlining the ROI as shown by figure 1(b).

IGES Volume Creation Steps for Rhino3D

In this section, the procedure on how to use Rhino3D to create a three-dimensional solid model from a layered two-dimensional IGES file is fully described. The user should be aware that if the Rhino3D version used does not already have the "IGES->Solids" toolbar (see fig. 1(c)), it can be imported. This can be done by picking the "Tools" menu and selecting "Toolbar Layout" and then "Import." The file "IGES Solids.WS" should be located and opened. Then "IGES->Solids" must be highlighted and "import" should be selected. The updated toolbar should appear in the upper right viewport (see fig. 1(c)).

To import the IGES file, select "Open" from the "File" menu, change the file type to "IGES (*.igs; *.iges)," and then select the file. The IGES file may have to be debugged and cleaned up. For instance, some curves may be too small or unneeded for the project. These curves must be removed by selecting them and then pressing the "delete" key. Also, there may be some curves that are crossed over with each other, which will cause later problems when expanding the curve into 3D. These curves must be broken up into individual curves. This is done by selecting the curve and typing "explode" on the command line. This command will break up the curve into individual line segments. Then, select the "none" option from "crossing line segments" and type "join" on the command line to create a curve. Do this for the other segments.

Solid Model Generation; Modeling Procedure and Construction Technique

The steps below illustrate the procedure on how to create a suitable three-dimensional solid model for processing in MSC/Patran using the IGES file developed from the ceramic composite specimen in the previous section.

Creating volumes from a two-dimensional curve.—
(1) From the imported IGES model, select the curve that should be converted into a solid. Then press button "1" in the "IGES Volume Creation" toolbar. Move the mouse to the center of the curve. When the word "Cen" appears next to the mouse pointer in the center of the curve, click the left mouse button. Figure 2 depicts the centered curve.

(2) Press button "2" in the toolbar. Four more shapes are created—two rectangles and two scaled smaller curves, see figure 3. If the new objects are not the correct size or spacing, refer to the table in appendix A for the Rhino commands to readjust the sizing and spacing. Select all of the new objects and the original curve (hold down [control] to select the individual elements or hold the left mouse button and drag over the area to select).

(3) Press button "3" in the toolbar. A loft line should appear stretched across the selected curves, figure 4(a). This loft line will create a surface by wrapping around the selected guide curves. Press [enter], a box will come up, then press [enter] again or select "ok." A surface will be created, figure 4(b).

(4) Press button "4" in the toolbar. Then, select just the two rectangles on the end as shown in figure 5. Hold down [control] to select the individual elements or hold the left mouse button, and drag over the area to select.

(5) Press button "5" on the toolbar. The rectangles will be turned into surfaces. Select the two surfaces as shown in figure 6. Hold down [control] to select the individual elements, or hold the left mouse button and drag over the area to select.

(6) Press button "6" on the toolbar. A solid surface is created. The command history will display whether the solid is error free or not. If there are errors, it is likely that the original curve was not centered

correctly. Select "undo" 15 times and redo the procedure from step 1. If all is well, proceed to the next curve and follow steps 1 through 6 again until all of the interior curves are converted in volumes. A rendered view of the newly created volume is shown in figure 7.

Creating cylinders to contain the volumes.—Certain volumes created will have too many sharp edges to work with in MSC/Patran. To fix this problem, cylinders can be created around the volumes to simplify the edges. The steps below describe how to perform this procedure with Rhino3D.

(1) Select an outside curve of the IGES model. Press button "7" on the toolbar and select the center of the outside curve so that the plane of the curve is placed on the x,y-plane as shown in figure 8. This action should also set the drawing mode to Planar—necessary in order for the cylinders to line up. If for some reason Planar is not on or was turned off (Planar mode is displayed on the bottom right of the Rhino window), you need to turn it on by typing "Planar."

(2) The next step is to create curves surrounding single or groups of close volumes. To do this, use the circle or ellipse buttons in the IGES->Solids panel. Only create curves around the volumes that are on the working plane (x,y-plane). Make sure there is enough space between the edges of the volume and the edge of the curve. Figure 9 shows a volume enclosed by a circle.

(3) Press button "8" on the toolbar. This will select all curves in the entire model so some curves will probably need to be deselected. By holding down [control], you can deselect curves. The curves to be highlighted are the ones just created on the x,y-plane. All other curves—usually only the outside outline curves—need to be deselected. See figure 10 for an example of this selection.

(4) The next step is to create the cylinders. Press button "9" on the toolbar and raise the cylinder with the mouse, using a front or side view, to just above the tops of the volumes as in figure 11. Then press button "9" again and create a cylinder just below the bottoms of the volumes. Finally, press [delete] to remove the curves.

(5) Select all of the cylinders just created. There is no easy way to do this because you do not want any of the volumes to be selected at this time. Remember to hold down [control] to select multiple items. Once all the cylinders are selected press button "10" on the toolbar. Then select all of the volumes on your working plane (the ones encompassed by the cylinders). They can be selected with a properly placed area select. Once the volumes are selected, press [enter]—the volumes will be removed from the cylinders. Repeat steps 1 through 5 until all volumes in the model have cylinders around them. Figure 12 shows the completed operation for a single slice.

Creating the exterior surface and combining with volumes.—This set of steps creates the outside outline surface between slices and then removes the cylinders from the end caps of the surface.

(1) Select two consecutive outside curves of the IGES file. Press button "11" on the "IGES Volume Creation" toolbar. This will create a loft line to make a surface between the curves. Ensure that the line created between the two curves is perpendicular by moving one of the points so that the line is straight in the front viewport and press [enter] when done (a window should pop up). Figure 13(a) shows a view of this. The surface created may look overly complicated. If so, select "Rebuild with x control points" and set x to a number that results in a relatively accurate representation of the curves, but without the excessive amount of control points (use the "Preview" button to view what the model will look like with x number of control points). Press [enter] or select "ok" when done. Figure 13(b) shows a surface between two slices that has an ideal number of control points. Repeat this procedure as necessary until the entire outside outline of the object is created.

(2) Press button "12" on the toolbar. This removes all curves, so there should only be the outside outline curves and nothing from the interior. If any were removed from the interior, it is likely that an interior entity was not created properly or finished and must to be repaired. Select "Undo" once and locate the object that wasn't created properly, then repair it using the techniques and tools outlined in appendix A.

(3) Select all of the cylinder halves outside of the model area and delete them. These should only be at the very ends of the model and are not needed for meshing, as shown by figures 14(a) and (b).

(4) Press button "13" on the toolbar. Select all the bottom circular curves of the cylinders on one side of the slice as depicted in figure 15(a). Then press [enter]. This will create cylinders that will ensure a proper, reliable Boolean operation later. Move the ends of the cylinders towards the closest end of the outside outline slice surface and extend it just past the end surface. Repeat this whole step for the other side of the slice. Figure 15(b) shows this step completed for one slice.

(5) Select the slice that you are working on and press button "14". This will turn the slice surface into a solid as shown in figure 16(a). Then select all of the cylinders (using the [control] key) just created in step 4 (the ones without the volumes inside) and press [enter]. This will remove the cylinders from the slice and create a hole for the cylinders with the volumes inside to sit in as shown in figure 16(b). Once this slice is complete, steps 4 and 5 need to be repeated on the other slices in the model. Figure 16(c) shows a rendered view of the completed transformation from the IGES curves into solids suitable for MSC/Patran.

(6) Finally, go to "File" and "Save As" and save the file as a <u>Parasolid file</u>, then select the default Parasolid type. A window may pop up stating to do that requires the operation "scale-convert units." This is all right as long as the units were set correctly in the Document properties (type "units" on the command line and change to the proper units—scaling should not be done unless you are converting). The reason for using the Parasolid file type is that it is fully supported in MSC/Patran, and it treats all objects as solids as opposed to a collection of surfaces, curves, and solids that the IGES-format export from Rhino3D does. However, it should be mentioned here that another option is available to generate an MSC/Patran file using an STL file format should the user elect not to generate a Parasolid file type. The STL is generally created via Rhino3D or any other image processing software. Translator software has been developed [1] to carry the transformation of an STL file format to MSC/Patran neutral file format. It is labeled as PNF translator. Appendix B gives a brief description of the code, and figure 19 represents a graphical user interface layout of the main page.

Actual Rhino3D Techniques and Tools Used

The table in appendix A describes each command in Rhino3D (ref. 6) used to create the solid using the buttons and also gives information about specific commands. Rhino3D can perform most all commands through the keyboard. The command line at the top of the screen displays the current and previous command data. To edit the command(s) associated with a button, hold down the shift key and right click the mouse. When entering a command, a space is considered the same as pressing [enter]. Also, the "!" command cancels all previous commands that may be in progress.

Generalized Summary of Volume Generation Techniques

The steps below review the procedure to recreate the IGES model as a true three-dimensional model.

(1) Replicate and reduce the size of a curve of a crack and/or void and place copies above and below the original. The z placement depends on the spacing between slices and estimation on how deep the cracks/voids go. This action is used to create a guide to follow for generating a surface.

(2) Create scaled rectangular end curves above and below the respective copies of the crack and/or void.

(3) Loft a surface around the five curves and cap the ends (rectangles).

(4) Join all parts of the surface to make a solid.

(5) Contain the newly created solid in two cylindrical solids through Boolean difference to make FE model meshing simpler (less sharp angles). The two cylinders meet in the middle of the newly created solid.

(6) Loft a capped surface between two of the outer edges of a slice. Connect the surfaces into a solid.

(7) Remove the shape of the cylinder(s), leaving the cylinder intact, from the slice solid using Boolean difference.

The steps used in Rhino3D are in some cases redundant in order to prevent potential problems with Boolean operations. Additionally, certain steps in Rhino3D could theoretically be combined into one step, but are not because of difficulties in automating the selection of specific objects.

A possible way to enhance this technique to be more accurate is to scan multiple planes of the object (i.e., x,y-plane and y,z-plane). After processing the sets of images in Velocity^2, the two IGES files can be compiled together into Rhino3D, and through the use of Boolean operations the two three-dimensional models combined into one. These Boolean operations would combine the volumes before the cylinders are made. Subsequently, the set of curves in the lofted, capped surface may be removed since interior detail is more desired for this analysis.

Application of the Volume Generation Techniques

The three-dimensional solid generated using Rhino3D was imported into the MSC/Patran program. The solid is then meshed resulting in 33 695 tetrahedral elements and 51 700 nodes, as shown in figure 17. The output depicted in this figure clearly indicates that the methodology applied is capable of modeling the structural abnormalities as they are detected by the NDE procedure. An FE stress analysis was also performed for the meshed solid, using the ANSYS general purpose software (ref. 8). The resulting von Mises stresses present in the analyzed ceramic composite are shown in figure 18. It is clearly noted that the stress concentration regions are aligned near the structural anomalies that were detected by the NDE technique applied and then modeled by the CAD–FE approach developed in this study.

Concluding Remarks

Prototyping a three-dimensional volume of a composite structure by means of reading a series of two-dimensional images generated via computed tomography (CT) and by using combined in-house and commercial software; e.g., Velocity^2, MSC/Patran, Rhinoceros 3D, and PNF was successfully demonstrated. A procedure that outlines all the steps to generate a structural model of a component that includes all anomalies detected by NDE imaging was developed and completed. Key major accomplishments are listed below:

(1) A CAD-based procedure to render a three-dimensional object from a series of CT scan images has been developed. Key elements detected by NDE such as manufacturing anomalies and structural deformities in a structure were modeled to match their actual shape as detected.

(2) The procedure applied in constructing a three-dimensional volume remains somewhat cumbersome. Automation, to reduce the number of steps involved in this procedure, is planned for the near future. Nevertheless, the current activities are promising, and structural assessment of a particular component is achievable.

(a)

Selected void to be modeled into three-dimensional volume

12.72 mm

2.53 mm

(b)

Negligible-size curve

IGES->Solids

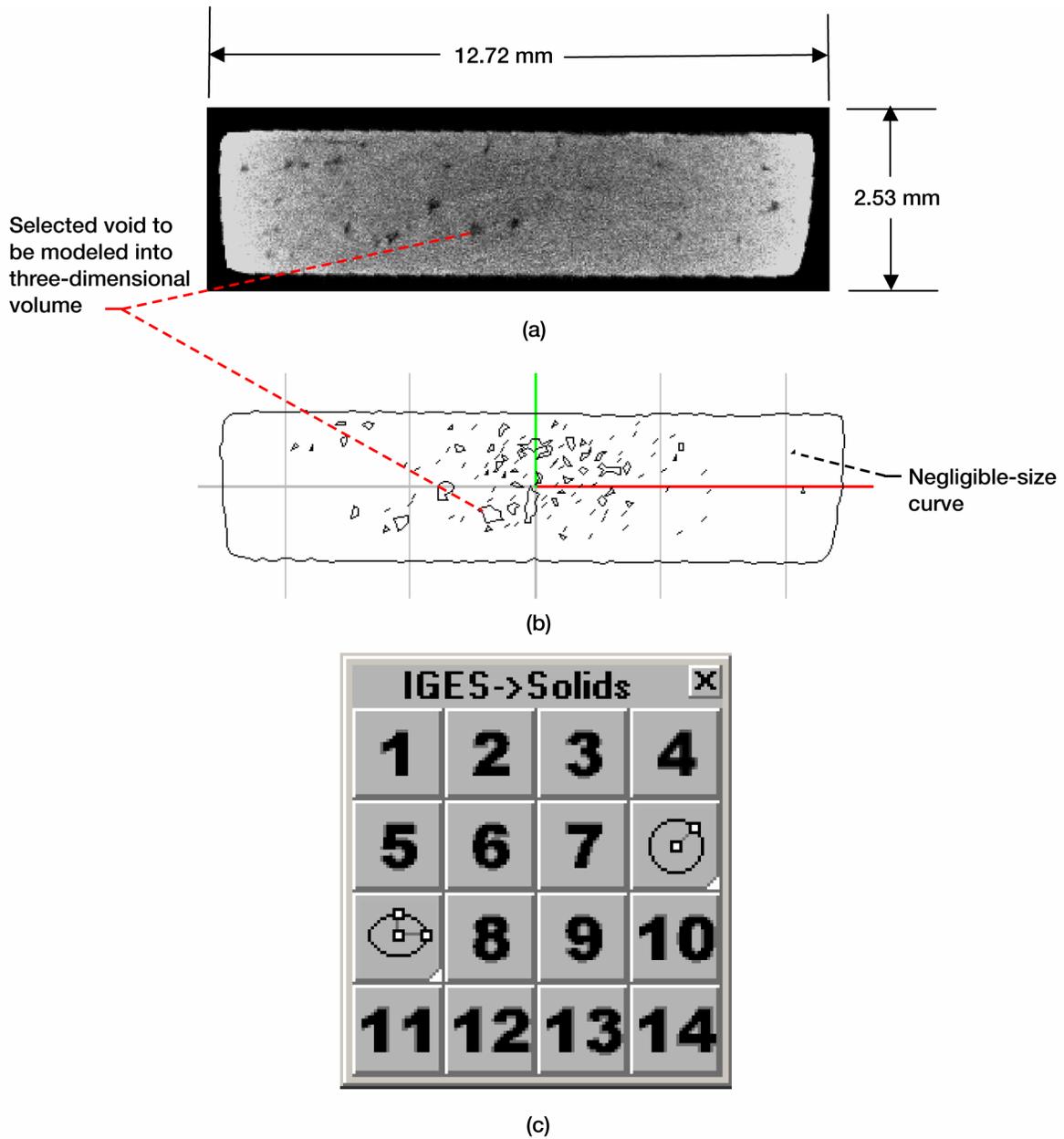| 1 | 2 | 3 | 4 |
5	6	7	
	8	9	10
11	12	13	14

(c)

Figure 1.—Background information for procedure to create three-dimensional geometry. (a) Computed tomography (CT) scan slice of composite material. (b) IGES representation of CT scan slice. (c) Custom-
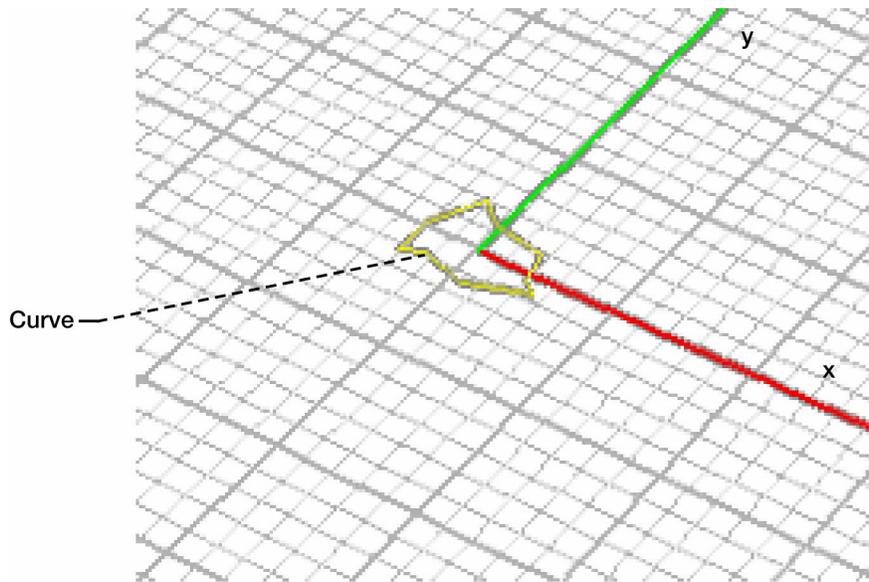
Figure 2.—Curve centered on x,y-plane at coordinates (0,0,0).

Figure 3.—Scaled curve and rectangular ends created in z space.
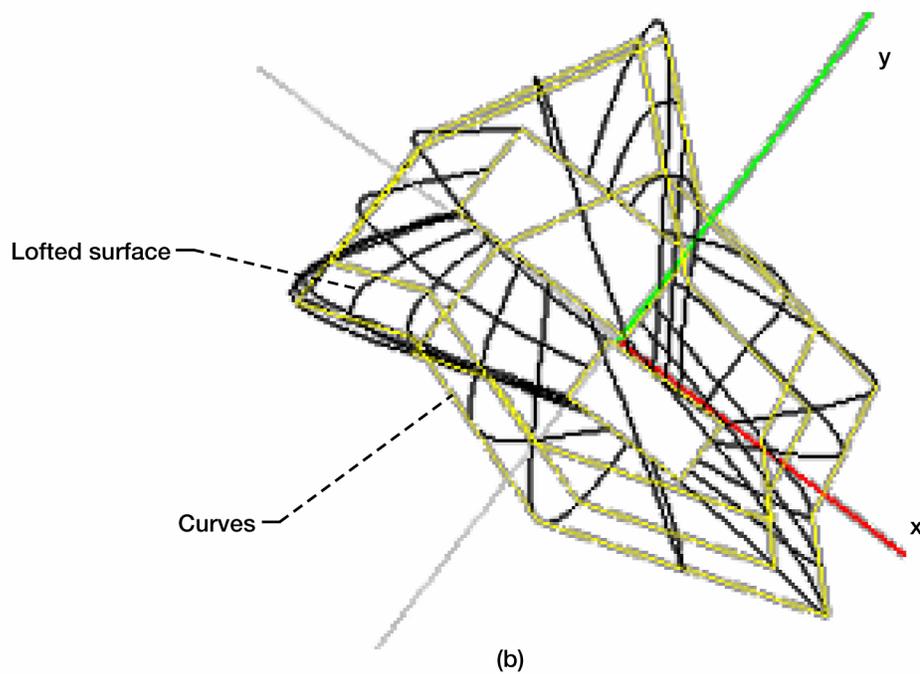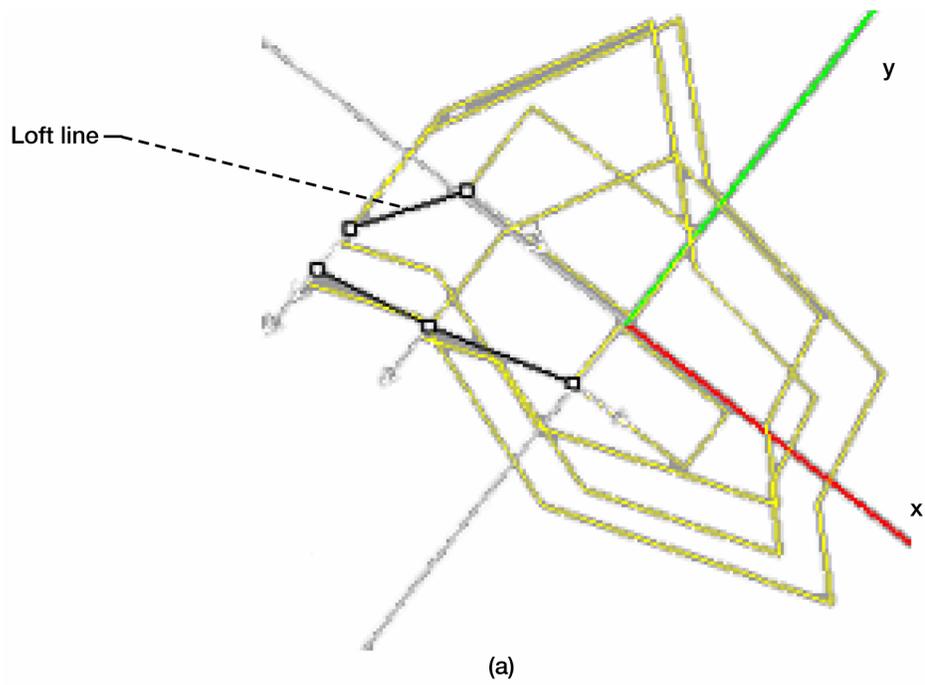
Figure 4.—Surface is lofted between the five curves. (a) Loft direction line between stretched curves. (b) Lofted surface around stacked curves.
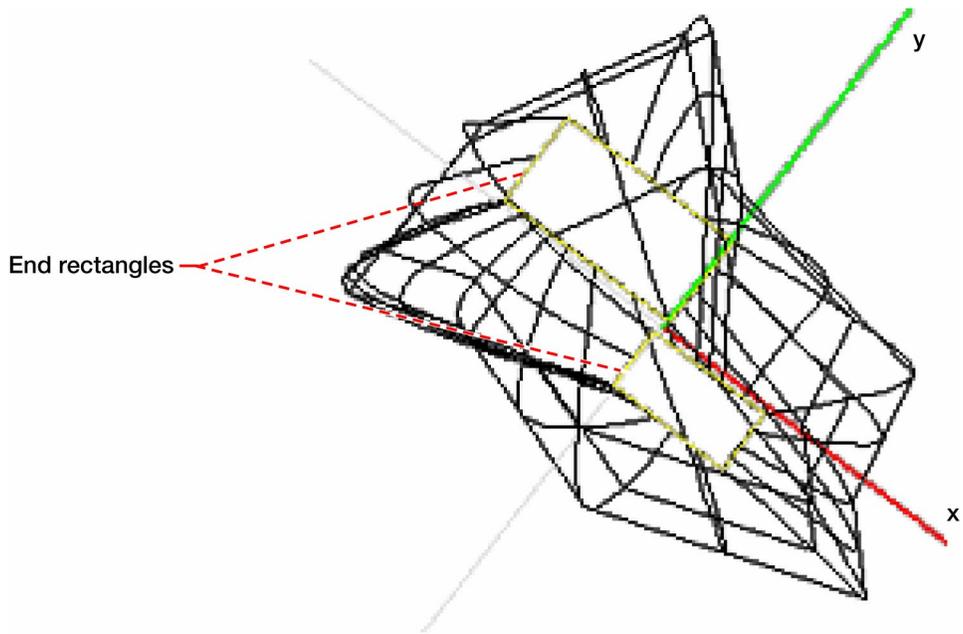
Figure 5.—End rectangles selected to create end caps.
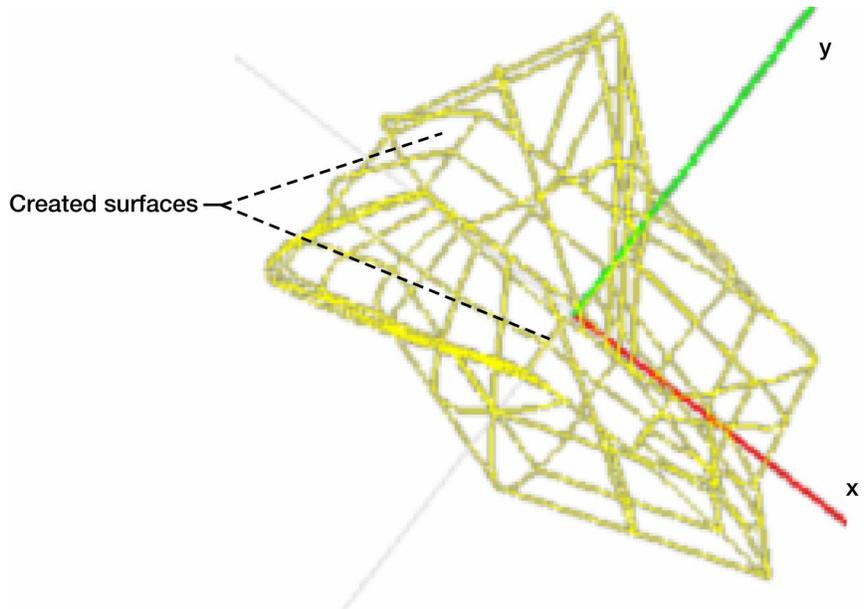
Figure 6.—Rectangles are made into surfaces for end caps.

Three-dimensional
modeled void ——- - - - - -
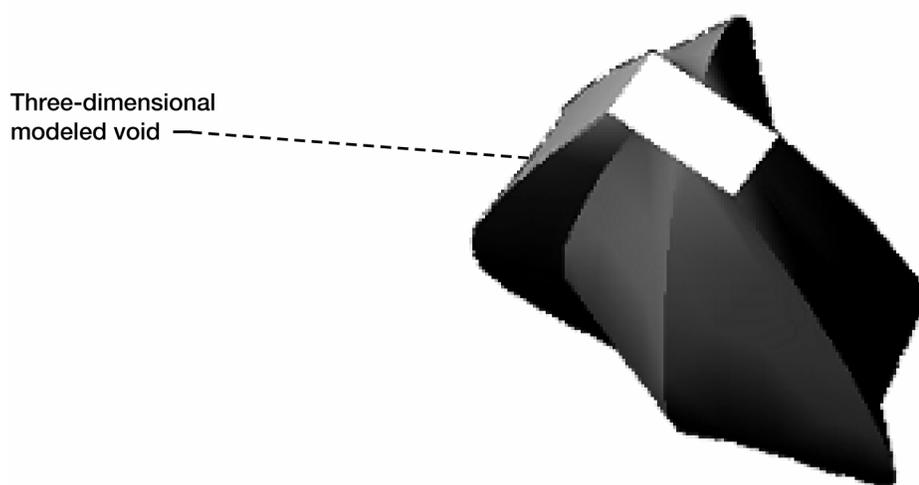
Figure 7.—Two surfaces are joined, creating solid. Rendered three-dimensional
view of solid is shown.

Outside outline curve —- - - -

Solid created
from previous steps —-
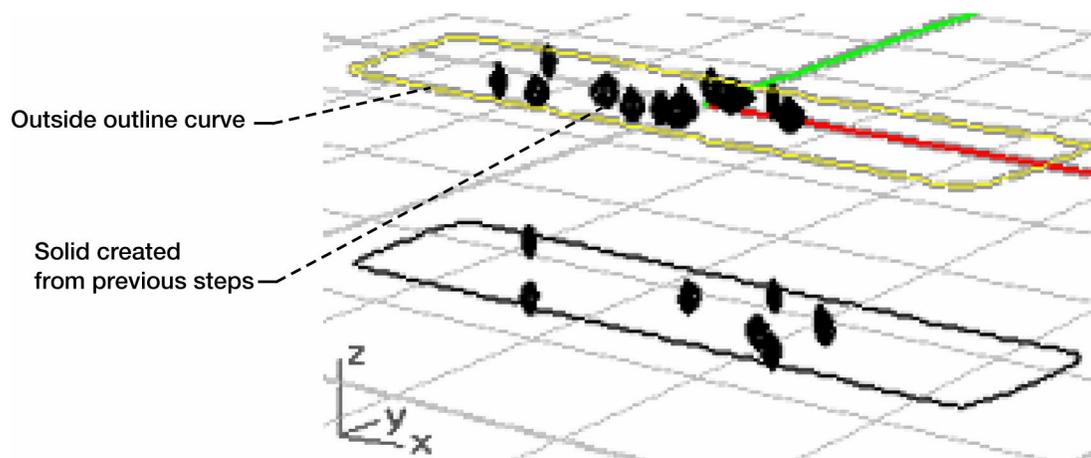
Figure 8.—Centered outside outline curve on x,y-plane at coordinates (0,0,0).
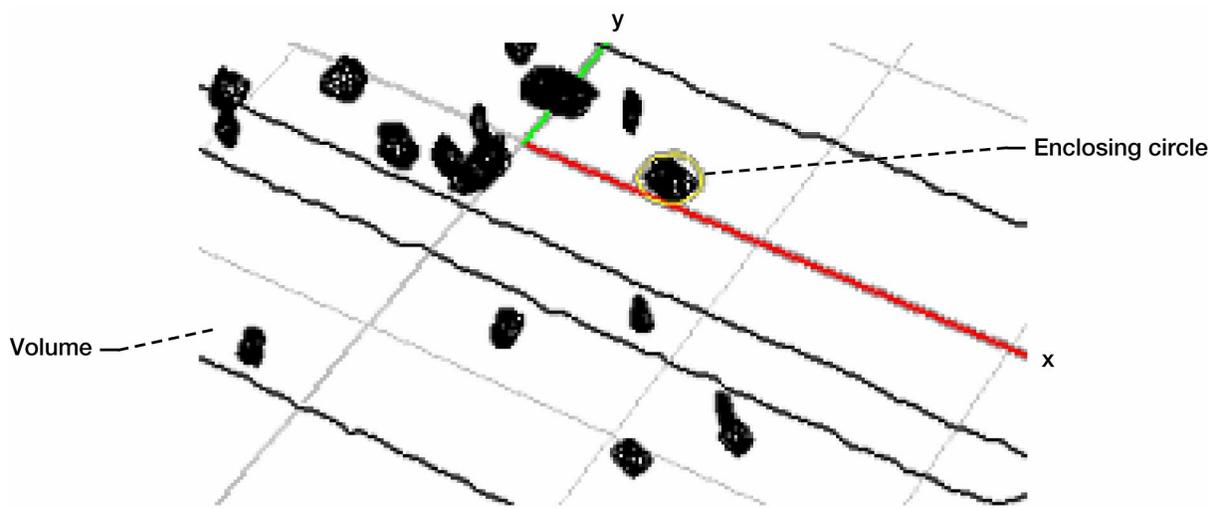
Figure 9.—Circle is made enclosing created volume.
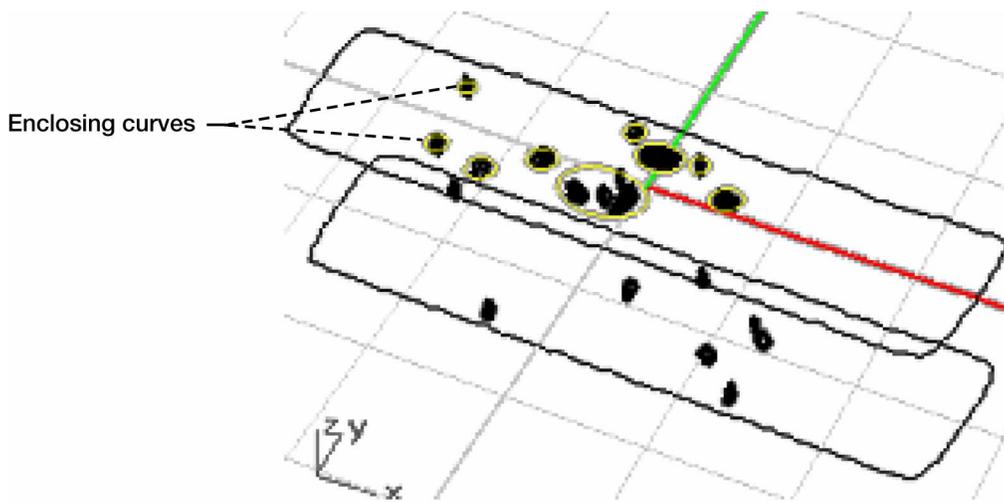
Figure 10.—Selected enclosing curves on working plane.
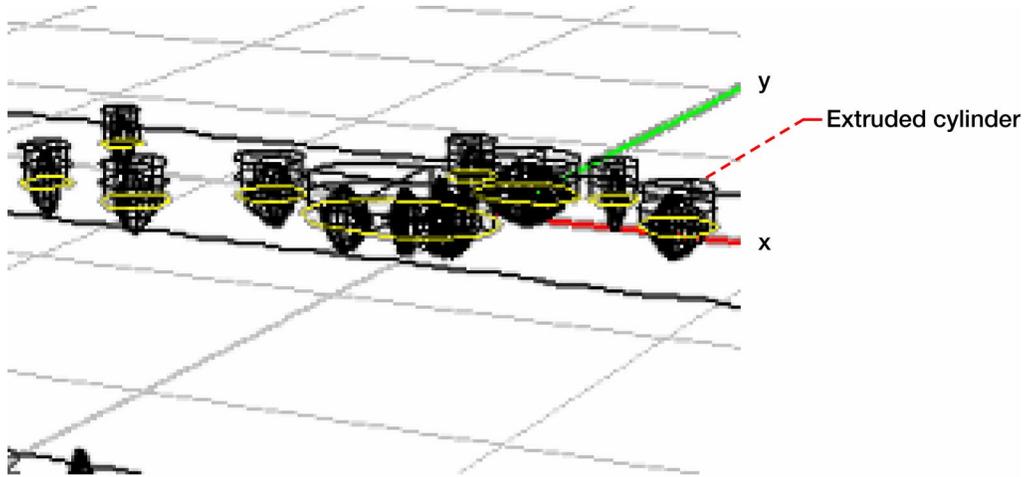
Figure 11.—Cylinders made by extruding curves into solid in both ±z directions. Cylinders in +z direction are shown.
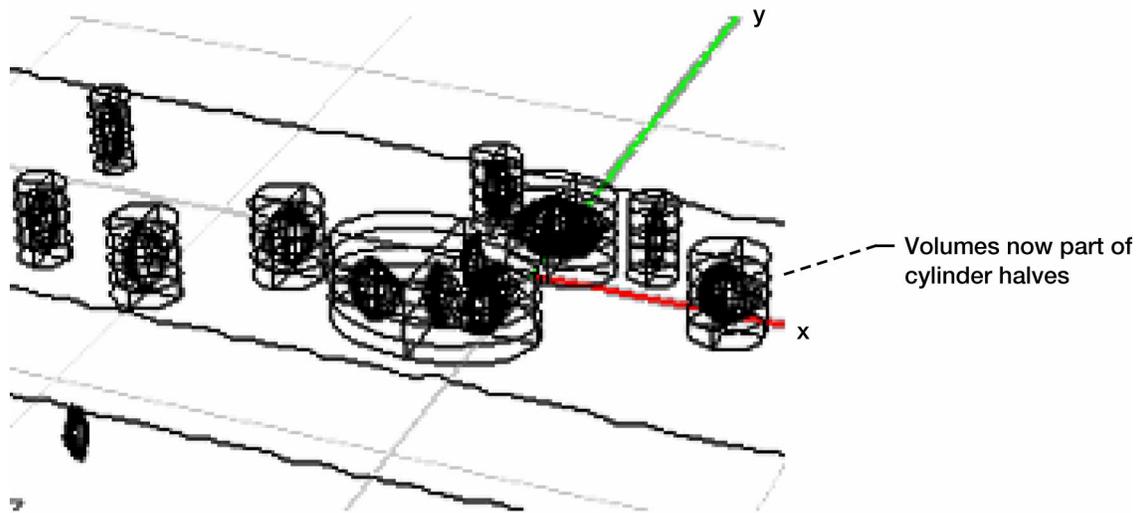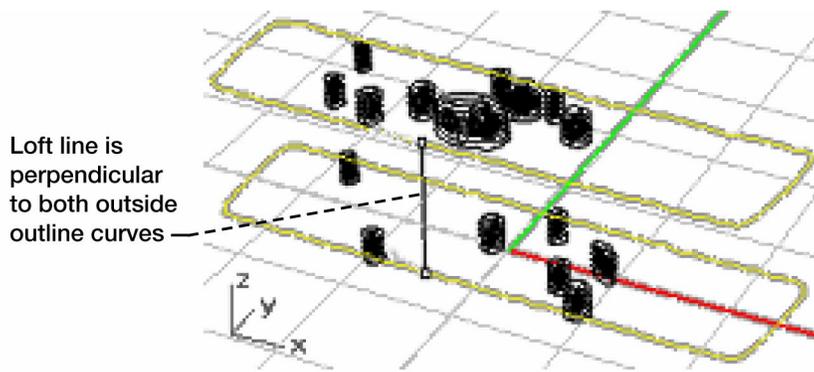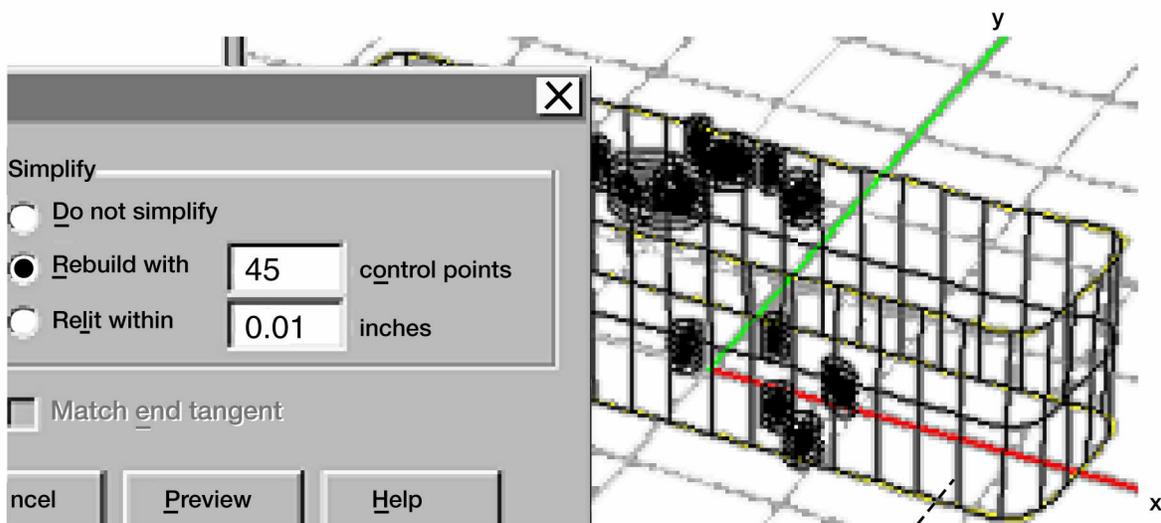

Figure 12.—Through Boolean operations, volumes are removed from cylinders.

Loft line is perpendicular to both outside outline curves —

(a)

Simplify

○ Do not simplify
● Rebuild with   45   control points
○ Relit within   0.01   inches

☐ Match end tangent

[ncel] [Preview] [Help]

Lofted surface is accurate to original
curves, but not overly complicated ⌐

(b)

Figure 13.—A surface is lofted around two outside outline curves. (a) Loft line is adjusted so it is perpendicular to two selected curves. (b) Control points are adjusted to reduce object complexity.
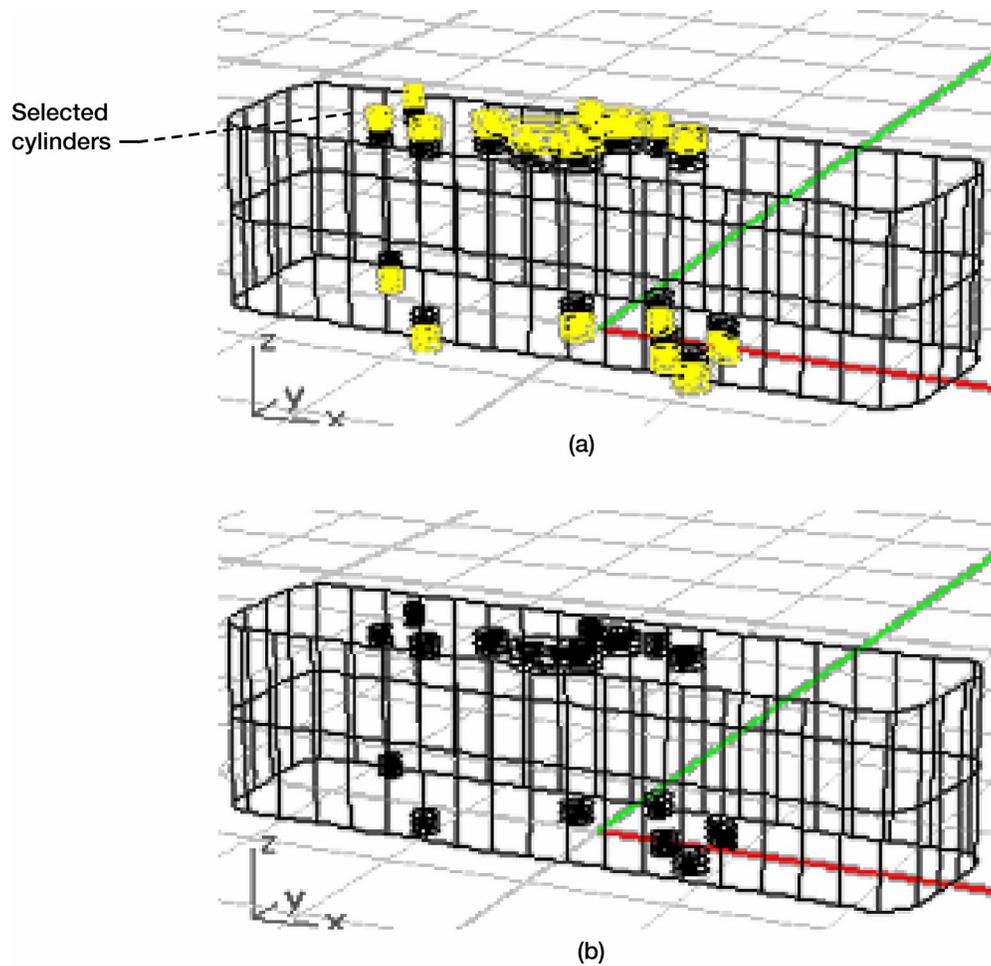
Selected
cylinders

(a)

(b)

Figure 14.—Unnecessary cylinder halves on extreme ends are removed. (a) Cylinder
halves on extreme end are selected; selected cylinders do not effect model
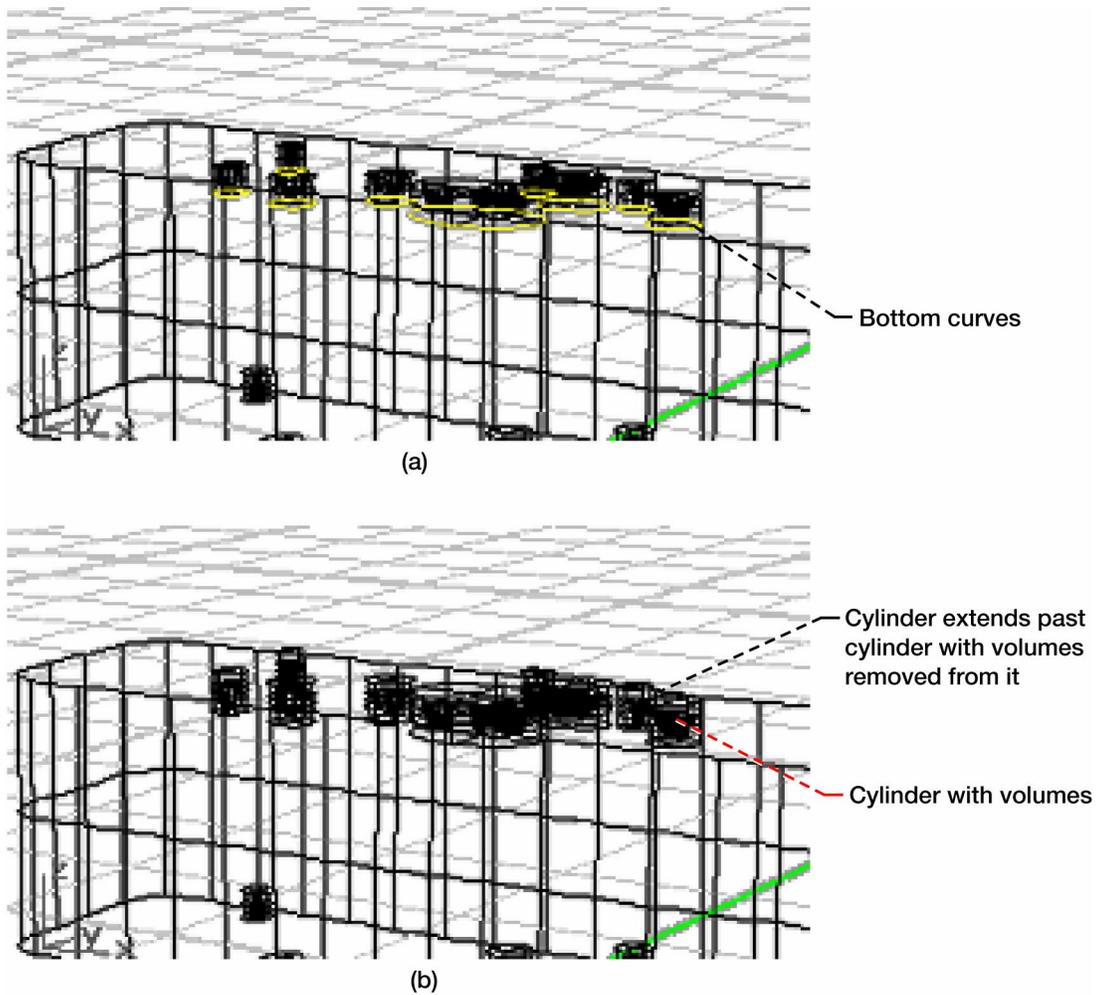because they are on ends. (b) Cylinders are deleted.

Figure 15.—New cylinders are created to reduce Boolean errors. (a) Curves that make up bottom
of cylinders are selected. (b) Curves from part (a) are extruded past cylinders to create
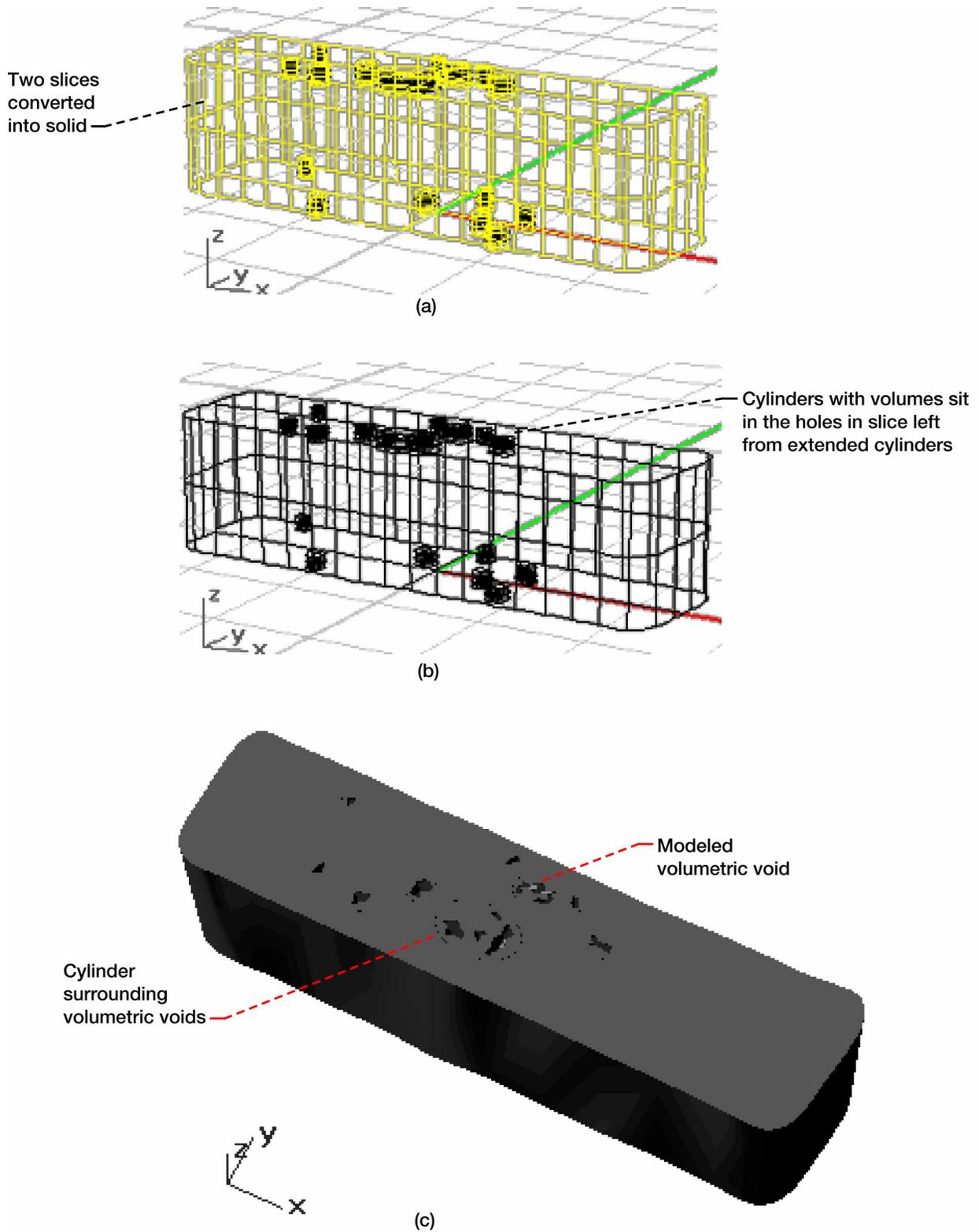overlapping cylinders.

Figure 16.—Two two-dimensional slices are converted into three-dimensional slice to be used in FE analysis. (a) Slices are selected and converted into solids. (b) Overlapping cylinders are also selected and then removed by Boolean operation from slice solids. (c) Rendered view of completed slice.
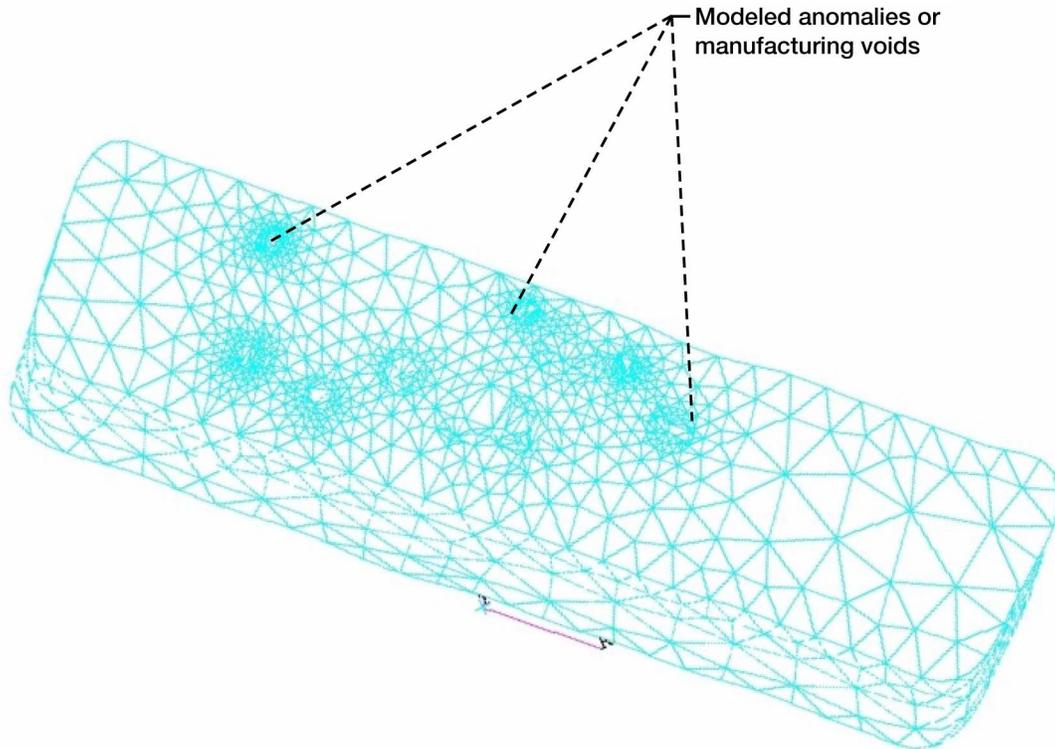
Modeled anomalies or manufacturing voids

Figure 17.—Three-dimensional finite element mesh generated from three-dimensional solid of figure 16.

Maximum stress location and anomalies site
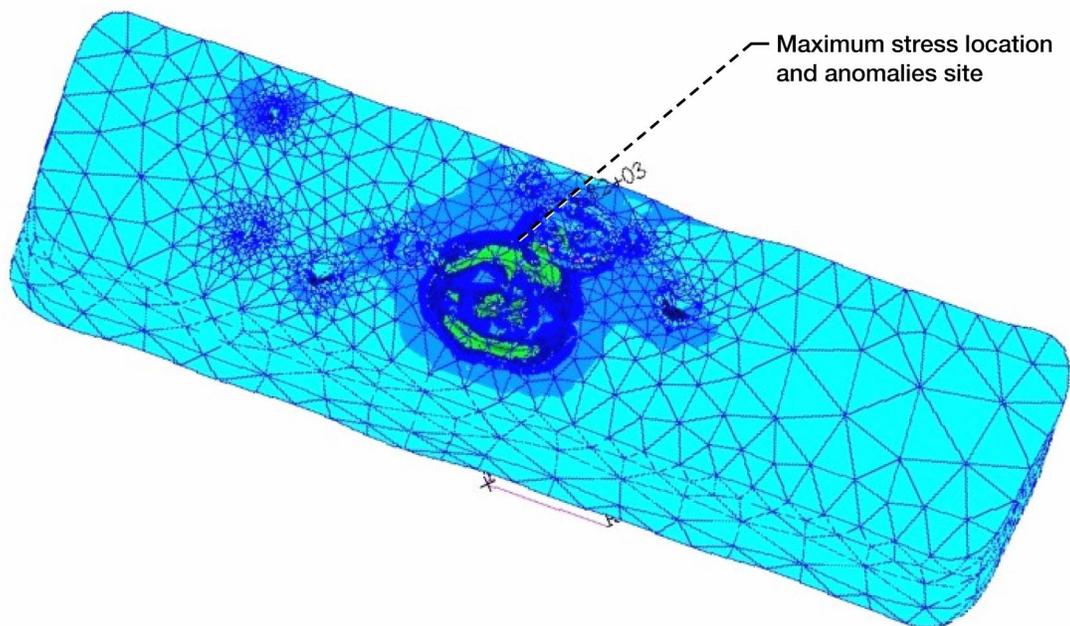
Figure 18.—Von Mises stress contours of solid with defects reconstructed from CT scans.

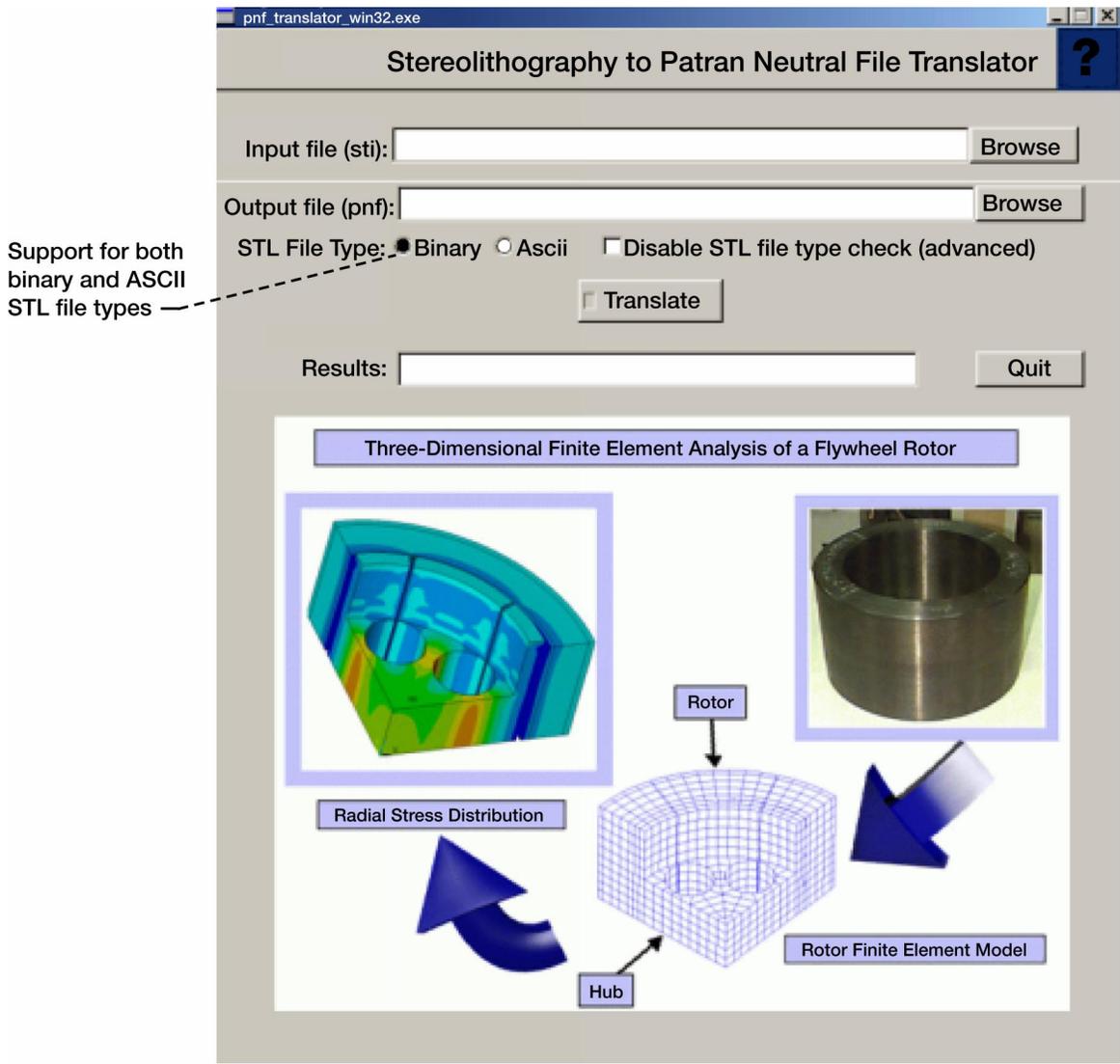Support for both binary and ASCII STL file types

Figure 19.—Window layout of PNF Translator graphical user interface.

Appendix A—Rhino3D Commands

The table below presents the commands used in Rhino3D to create a three-dimensional
solid able to be processed in MSC/Patran for FE model generation.

RHINO3D TOOLBAR COMMANDS AND DESCRIPTIONS

Button	Command	Description
1	! C Plane Origin All cen	Moves origin of document to center of selected object (after user selects center of object). The "cen" command makes cursor snap to center of object.
2	! Zoom Selected All	Zooms in all viewports to selected object.
	Copy Clip	Copies curve to clipboard.
	Paste	Pastes object from clipboard to original location.
	Scale2D 0, 0, 0 .75 or Scale2D \<origin> \<factor>	Scales curve equally in all two-dimensional directions. The 3 numbers immediately following the command set origin of scale. The last number sets scale factor. Scale factor may need to be changed depending on proportions of model.
	Copy Clip	Copies newly scaled curve
	Move 0, 0, 0, 0, 0, 0.05 or Move \<origin> \<destination>	Moves scaled curve to above original curve. First set of numbers state where curve is to be moved from and second set of numbers state destination of curve. The z number from second (destination) set is only one that may need changed depending on proportions of model (always positive).
	Paste	Pastes scaled curve at origin.
	Move 0, 0, 0, 0, 0, –0.05 or Move \<origin> \<destination>	Moves scaled curve to below original curve. First set of numbers state where curve is to be moved from and second set of numbers state destination of curve. The z number from second (destination) set is only one that may need to be changed depending on proportions of model (always negative).
	Rectangle –0.0125, –0.00625, 0.1, 0.0125, 0.00625, 0.1 or Rectangle \<lower_left> \<upper_right>	Creates rectangle above original curve as well as scaled curve. The z position and size of rectangle may need to be varied depending on proportions of model. Size of rectangle is determined to be lower left and upper right points. The z number should always be positive.
	Rectangle –0.0125, –0.00625, –0.1, 0.0125, 0.00625, –0.1 or Rectangle \<lower_left> \<upper_right>	Creates rectangle below original curve as well as scaled curve. The z position and size of rectangle may need to be varied depending on proportions of model. Size of rectangle is determined be lower left and upper right points. The z number should always be negative.
3	! Loft	Creates surface by wrapping around selected curves. If loft does not look right, original curve may have not been centered at origin correctly, and it will need to be undone and started over.
4	! Delete	Deletes curves leaving behind lofted polysurface.
	Rectangle –0.0125, –0.00625, 0.1, 0.0125, 0.00625, 0.1 or Rectangle \<lower_left> \<upper_right>	Recreates rectangle from button two. Values after "Rectangle" should be same as positive z value in button two.
	Rectangle –0.0125, –0.00625, –0.1, 0.0125, 0.00625, –0.1 or Rectangle \<lower_left> \<upper_right>	Recreates rectangle from button two. Values after "Rectangle" should be same as negative z value in button two.
	SelNone	Ensures nothing is selected.
5	! PlanarSrf	Creates two rectangular surfaces from two rectangular curves.
	Delete	Removes two rectangular curves.
6	! Join	After all three surfaces are selected (lofted surface and two rectangles), combines surfaces into one polysurface.
	Check	This checks polysurface for errors. If errors are detected, object needs to be rebuilt. You may have to undo to original curve to fix errors.

7	! Planar	Forces new objects to be drawn on x,y-plane.
	Cplane Origin All Cen	Sets outside outline selected to be on x,y-plane and with "Planar" command, requires all new objects to originate from same plane. After this, circles and ellipses are created surrounding volumes.
8	! SelCrv	This selects all curves in model. This should only be outside outline curves. If interior object curve is selected, it is likely that interior object needs to be reconstructed.
9	! Extrude	This extrudes the curves around the volumes to create a cylinder. The curve is extruded twice: once for the top half of the volume and once for the bottom half of the volume.
10	! Boolean Difference Delete Input=Yes	This function removes the volumes from the cylinders as well as deletes the volumes.
11	! Loft	Once outside outline curves are selected, this will create a surface between the curves. If the surface appears to be overly complex, select the option to rebuild with a specific number of control points and select a number that makes a good approximation (use the preview button to view the changes).
12	! SelCrv	This selects all curves in the model. This should only be the outside outline curves. If an interior object curve is selected, it is likely that the interior object was not constructed correctly and needs to be reconstructed.
	Delete	Deletes all selected curves.
13	! Extrude	Creates additional cylinders that will ensure a correct Boolean operation. The cylinders are created from the bottom edge curve of the cylinders that contain the volume entities.
14	! Cap	Puts end caps on the outside outline surface.
	Boolean Difference Delete Input=Yes	Removes the cylinders from the outside outline surface so that the other cylinders with the volume entities will fit. If you suspect there are errors in a surface, select that surface and type "Check." If errors are reported, you need to redo that surface so that it is error free.

Appendix B—MSC/PATRAN Neutral File Generation

As part of the work to convert the CT scanned images into an FE solid model a computer program was written to translate the Velocity^2 stereolithography (STL) files into an IGES format readable by MSC/Patran used for mesh generations.

Velocity^2 can provide detailed three-dimensional geometry of the exterior of an object through interpolation between image slices. This geometry, which is polygonized three-dimensional data, is written in the common STL file format. However, this format is not suitable to be read by MSC/Patran. Therefore, a software translator to convert the STL file into a neutral file compatible with MSC/Patran was developed. The code is written in the C language, and it can recognize both ASCII and binary formats of the STL file. The code is identified as PNF translator where PNF stands for "Patran neutral file." The code was complied to be used on the Silicon Graphics IRIX platform. However, using the FLTK C++ libraries (ref. 9), the translator was updated to run in any window environment in addition to enhancing its error prevention and detection. Figure 19 shows the newly developed graphical user interface window layout displaying all the latest enhancements implemented.

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13. ABSTRACT (Maximum 200 words)

Nondestructive evaluation (NDE) is one of several technologies applied at NASA Glenn Research Center to determine atypical deformities, cracks, and other anomalies experienced by structural components. NDE consists of applying high-quality imaging techniques (such as x-ray imaging and computed tomography (CT)) to discover hidden manufactured flaws in a structure. Efforts are in progress to integrate NDE with the finite element (FE) computational method to perform detailed structural analysis of a given component. This report presents the core outlines for an in-house technical procedure that incorporates this combined NDE-FE interrelation. An example is presented to demonstrate the applicability of this analytical procedure. FE analysis of a test specimen is performed, and the resulting von Mises stresses and the stress concentrations near the anomalies are observed, which indicates the fidelity of the procedure. Additional information elaborating on the steps needed to perform such an analysis is clearly presented in the form of mini step-by-step guidelines.

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