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MEMORANDUM

To: Technology Development Program Manager, Dam Safety Office
Attn: 84-44000(LKrosley)

From: Jonathan East, Civil (Structural) Engineer/s/
Waterways & Concrete Dams Group 1 (86-68110)

Subject: Transmittal of Dam Safety Technology Development Report DSO-2018-07- Finite Element Model Development with LS-PrePost.

Attached for your use is the DSO-2018-07-Finite Element Model Development with LS-PrePost report, that has been prepared by the Technical Service Center at the request of the Dam Safety Office. The report will be available in Adobe Acrobat Format on the Dam Safety website and will be loaded into DSDAMS.

If you have any questions, please contact me at 303-445-3217 or via email at JEast@usbr.gov

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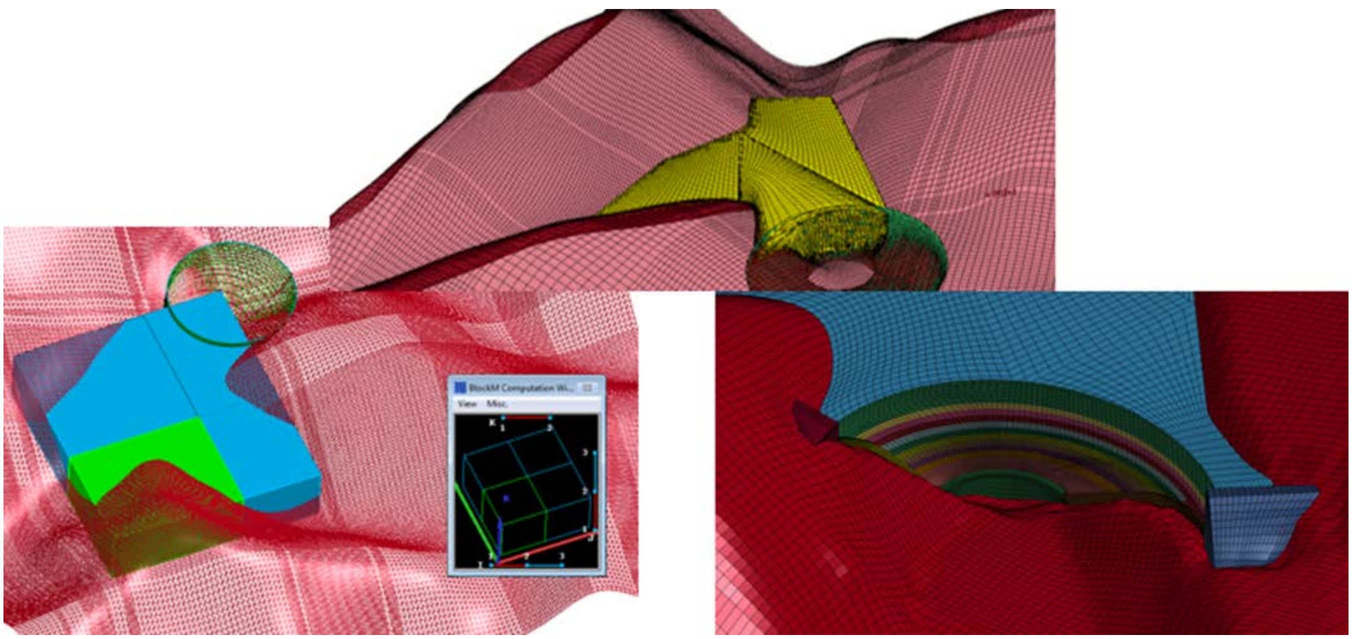
RECLAMATION

Managing Water in the West

Finite Element Model Development with LS-PrePost

DSO-2018-07

Dam Safety Technology Development Program



U.S. Department of the Interior
Bureau of Reclamation
Technical Service Center
Denver, Colorado

May 2018

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REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

T1. REPORT DATE: MAY 2018	**T2. REPORT TYPE:** TECHNICAL	**T3. DATES COVERED** September 2016 – May 2018

T4. TITLE AND SUBTITLE
Finite Element Model Development with LS-PrePost

5a. CONTRACT NUMBER

5b. GRANT NUMBER

5c. PROGRAM ELEMENT NUMBER

6. AUTHOR(S)
Jonathan East, P.E, Denver, CO (BOR, 303 445 3217)

5d. PROJECT NUMBER
DSO-2018-07

5e. TASK NUMBER

5f. WORK UNIT NUMBER
DMFEA

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
Department of the Interior, Bureau of Reclamation
Technical Service Center, Denver Federal Center
PO Box 25007 (86-68110)
Denver, CO 80225-0007

8. PERFORMING ORGANIZATION REPORT NUMBER

9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES)
Department of the Interior, Bureau of Reclamation
Dam Safety Office, Denver Federal Center
PO Box 25007 (84-44000)
Denver, CO 80225-0007

10. SPONSOR/MONITOR'S ACRONYM(S)
BOR/USBR: Bureau of Reclamation
DOI: Department of the Interior
DSO: Dam Safety Office

11. SPONSOR/MONITOR'S REPORT NUMBER(S)
DSO-2018-07

12. DISTRIBUTION / AVAILABILITY STATEMENT

13. SUPPLEMENTARY NOTES

14. ABSTRACT: Currently in the TSC, nonlinear finite element analyses (FEA) of concrete dams are completed to better understand the nonlinear behavior of such structures. The process of FE model development for a nonlinear model is very labor intensive and time consuming; some projects last multiple years and absorb Reclamation's limited personnel resources.

The quality of the finite element (FE) mesh is a key factor affecting the accuracy of FE analysis. This research investigates a new method of developing FE models using 3D AutoCAD and LSDYNA PrePost (LS-PrePost) as an alternative program to use. The tasks included in this research are to develop a new procedure for building FE models using 3D AutoCAD and LS-PrePost, develop a procedure for inputting the terrain topography into FE models using AutoCAD Civil 3D, document the modeling procedures of using AutoCAD and LS-PrePost in a report, evaluate the ability for local mesh refinement, and describe the model self-check capacity using LS-PrePost.

15. SUBJECT TERMS :

16. SECURITY CLASSIFICATION OF: UU			**17. LIMITATION OF ABSTRACT:** SAR	**18. NUMBER OF PAGES** 63 (not including Appendix)	**19a. NAME OF RESPONSIBLE PERSON:** Jonathan East
a. REPORT UU	**b. ABSTRACT** UU	**c. THIS PAGE** UU			**19b. TELEPHONE NUMBER** 303 445 3217

S Standard Form 298 (Rev. 8/98)
P Prescribed by ANSI Std. 239-18

Bureau of Reclamation

Dam Safety Technology Development Program

Waterways & Concrete Dams Group 1

DSO-2018-07

Finite Element Model Development with LS-PrePost

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Acronyms and Abbreviations

CAD	computer aided drafting
FE	finite element
FEA	finite element analysis
LS-PrePost	LSDYNA PrePost
TSC	Technical Service Center

Keywords

Finite element analysis, finite element modeling, finite element model development, concrete dam structural analysis, concrete dam finite element models.

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Abstract

The finite element analyses (FEA) of concrete dams are performed by the Technical Service Center (TSC) to better understand the nonlinear behavior of such structures. The process of finite element (FE) model development for a dam-reservoir-foundation system is very labor intensive and time consuming; some projects last multiple months and absorb Reclamation's limited personnel resources.

The quality of the FE mesh is a key factor affecting the accuracy of FE analysis results. Currently, Truegrid software is exclusively used by the Waterways and Concrete Dams Group 1 to generate the mesh for FE models of concrete dams for analysis using FE software LS-DYNA. Truegrid uses a text based input file with a limited graphical user interface to generate FE mesh models as an input for LS-DYNA.

This research investigates a new approach of developing FE models using 3D AutoCAD and LSDYNA PrePost (LS-PrePost) as an alternative to the current practice. The tasks included in this research are:

- Develop a new procedure of building a FE model using 3D AutoCAD and LS- PrePost.
- Develop a procedure for inputting the terrain topography in to FE model using AutoCAD Civil 3D.
- Document the modeling procedures of using AutoCAD and LS-PrePost in a report for others to utilize. This report satisfies this task.
- Evaluate the ability for local refinement of mesh.
- Describe the self-check capacity for models using LS-PrePost.

Background

Ongoing FE analysis projects, including issue evaluations for Deadwood and C.C. Cragin Dams, have utilized AutoCAD modeling to assist in FE modeling the structure for small aspects of each project, showing promise that significant modeling time may be saved with the use of computer aided drafting (CAD) programs. Due to project budgets, schedules, and unfamiliarity with modeling in AutoCAD and LS-PrePost, significant development of models has not been performed using either program. Although minor progress has been made, the abilities of using the full capacity of CAD programs for FE modeling is limited, therefore this research was initiated to further and more adequately understand and assess if using CAD programs in FE modeling would be a more efficient use of budget and resources than current practice.

The research documented in this report addresses the potential of using AutoCAD to develop 3D models that can be meshed in LS-PrePost for FE analysis. All of Reclamation’s current contract drawings are generated in AutoCAD or other Autodesk software, so most engineers are familiar with AutoCAD, reducing the learning curve for developing models for FE.

In addition, AutoCAD Civil 3D can be used to create topography surfaces, which can be used in FE models. Terrain surfaces in Civil 3D can be modified to provide as much detail as required and can be developed using USGS maps, existing drawings, or survey data. Once the terrain surface is created, it can be incorporated into the FE model providing a high level of detail and accuracy typically required for analysis of a dam project. Figure 1 shows an example of an AutoCAD Civil 3D surface model with a 3D arch dam. This research report will document incorporating Civil 3D terrain models into FEA models.

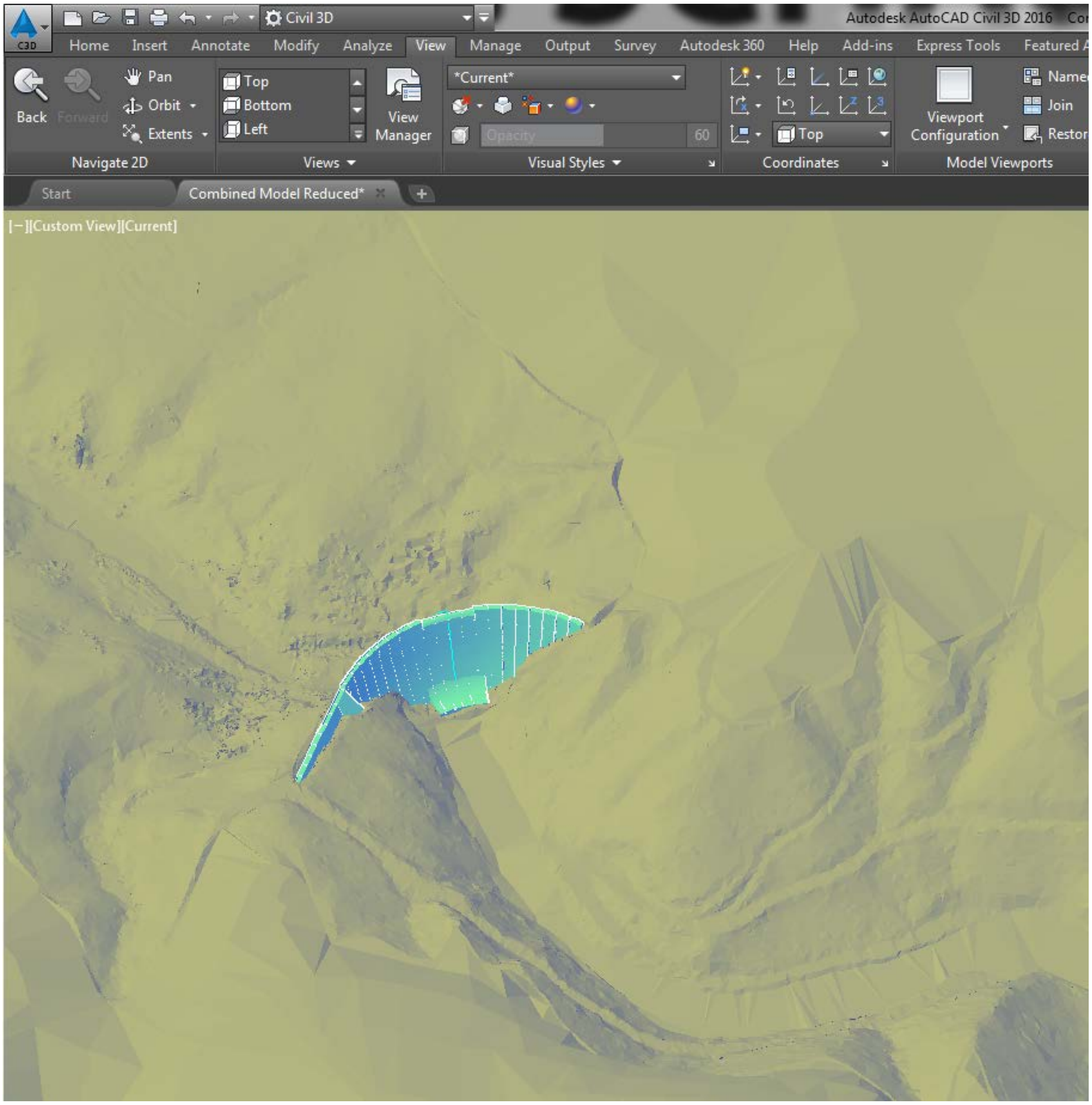


Figure 1 - Topography of an arch dam in AutoCAD Civil 3D (reservoir not shown).

In addition to terrain modeling, AutoCAD can be used to develop 3D models from simple to complex geometrical shapes that can be imported directly into FE programs. Figure 1 and Figure 2 show a 3D object, a monolith of a gravity dam, generated in AutoCAD. Figure 3 shows the dam monolith imported into LS-PrePost and meshed for FEA using the geometry built in AutoCAD.

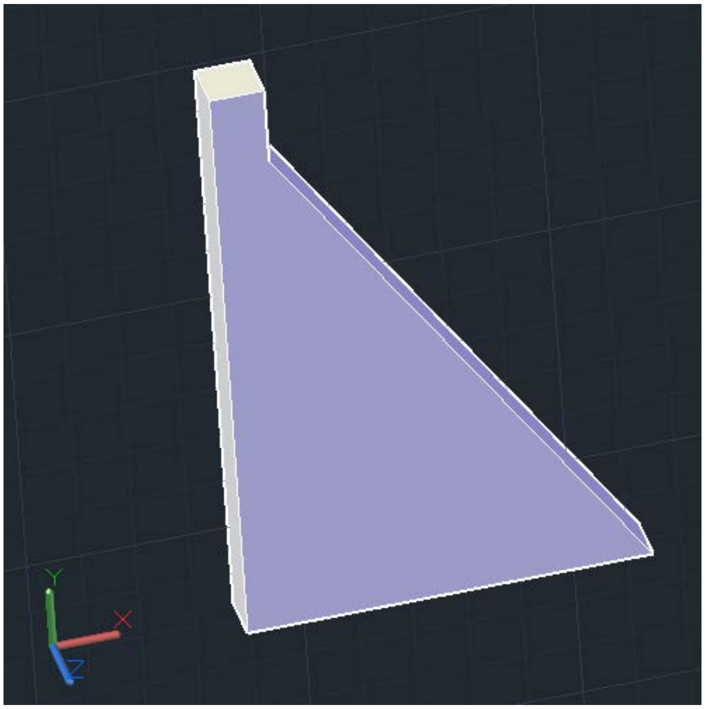


Figure 2 - 3D gravity dam model in AutoCAD.

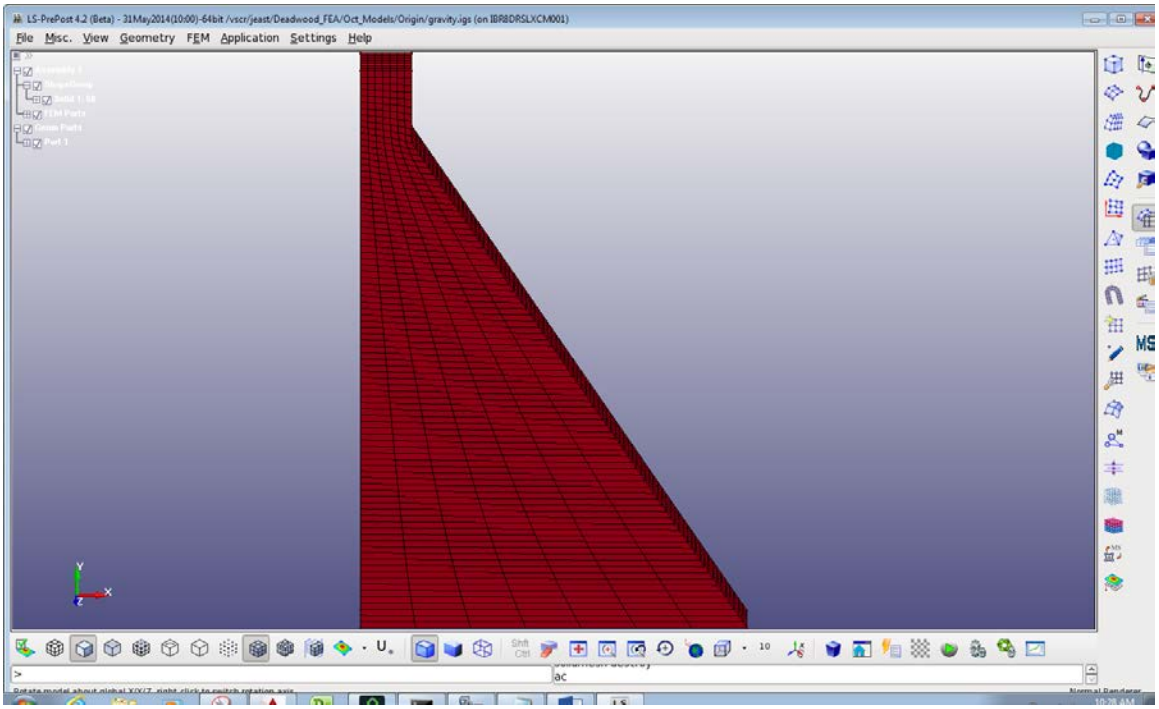


Figure 3 - AutoCAD model (above) and meshed model in LS-PrePost.

LS-PrePost can automatically generate FE meshes. The ability to automatically generate different mesh type, hexahedral or tetrahedral for example, may be used to optimize models based on the level of detail required. For example, a finer mesh may be required near a stress concentration, but may not be required everywhere. The automatic meshing in LS-PrePost can automatically mesh the area with the stress concentration within specified parameters, then transition to a coarser mesh where the fine level of detail is not required. An example of automatically meshing near a stress concentration is shown in Figure 4. However, it is important to note that mesh refinements, similar to what are shown below, require more time to ensure that nodes are properly connected.

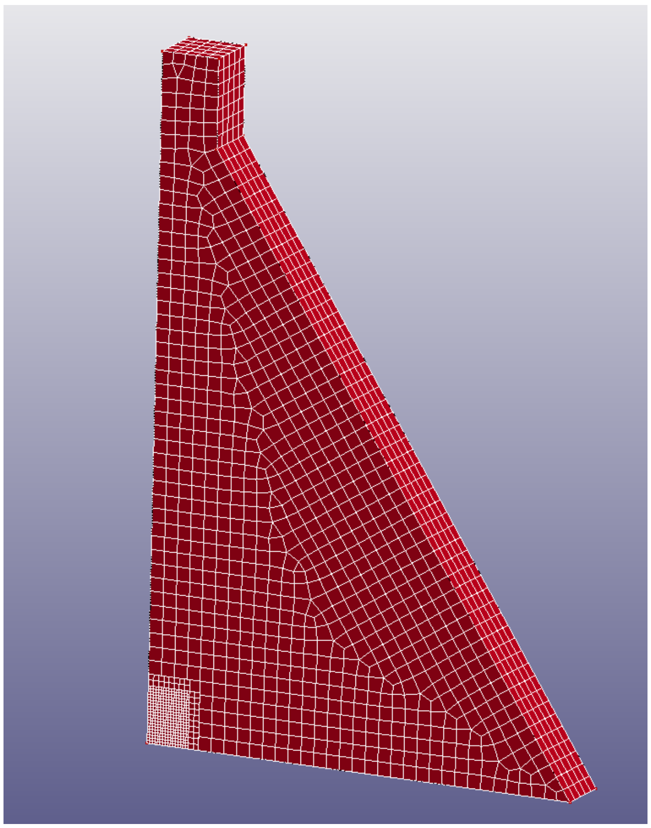


Figure 4 - Automatic mesh at stress concentration at the dam heel.

While this report is intended to provide a resource on how to create a model using AutoCAD and LS-PrePost, it is however not intended to be a directive on how a model should be built. In this report, the model development is covered for two types of models: a basic and an advanced model. The development of a simplified model of a stilling basin, called the "Basic Model." The development of the Basic Model will be treated as a tutorial on how to use the LS-PrePost software to develop an FE model from an AutoCAD model, including preparing the model for running in LS-DYNA. Following the Basic Model, an "Advanced Model" was developed, consisting of an arch dam, foundation and reservoir. The Advanced Model is more representative of a full dam model. Similar to the Basic Model, the Advanced Model acts as a tutorial for different techniques than those developed during the Basic Model. Both the Basic and Advanced Models use shapes or curves imported from AutoCAD, however, it should be recognized that the 3D Modeling program used to develop the shape models to mesh in LS-

PrePost is arbitrary; the shapes imported are the generic .iges file format and are not proprietary to Autodesk.

Generating and modifying shapes in LS-PrePost is only briefly mentioned below in this tutorial, but the mechanics of building an FE model in LS-PrePost are similar to building geometry in LS-PrePost so a user would be able to develop shape modeling abilities if desired based on this tutorial.

Prior to proceeding on any FE model, it is important to have a background of FEA. Discussion of FEA, however, is not the scope of this report, for fundamental concepts of intro to FEA background that including discretization, element types, mesh types, and boundary conditions in a FE model, readers are referred to introductory FEA books or publications developed by Salamon et al [1,2,3].

Finite Element Modeling Software

Autodesk Drafting Software

CAD and AutoCAD in general have become industry standards for drafting designs. In the past, designs were drawn and displayed as 2D representations of the design, yet as computer technology has developed, more designs are being completed as 3D models. Newer Autodesk software programs are adapting to this, where a design built as a 3D model and 2D sections are then cut from the model, which is done in Revit or Inventor [4,5]. In addition to structural models, Autodesk Civil 3D can build terrain surfaces from survey data or existing topography maps that are interactive and allow the user to grade sections along alignments or grade to a specified elevation or at a specific slope. With all of these modeling tools, designs are transitioning to full 3D models instead of 2D representations. Figure 5 and Figure 6 show AutoCAD and Civil 3D models, respectively.

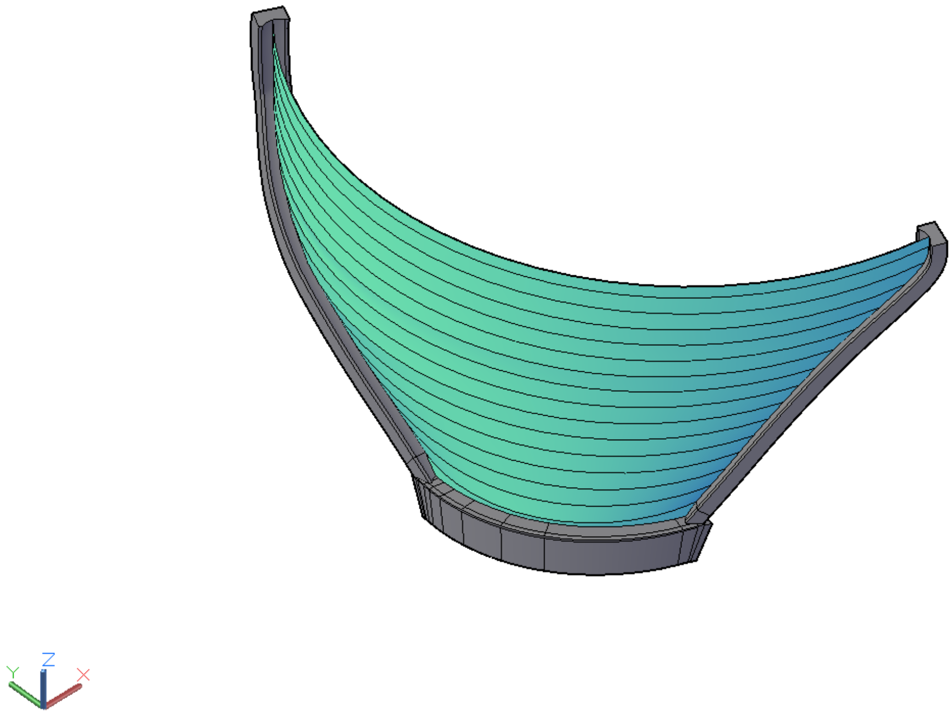


Figure 5 - Example of an AutoCAD 3D model of an arch dam (shell) and foundation blocks (solid).

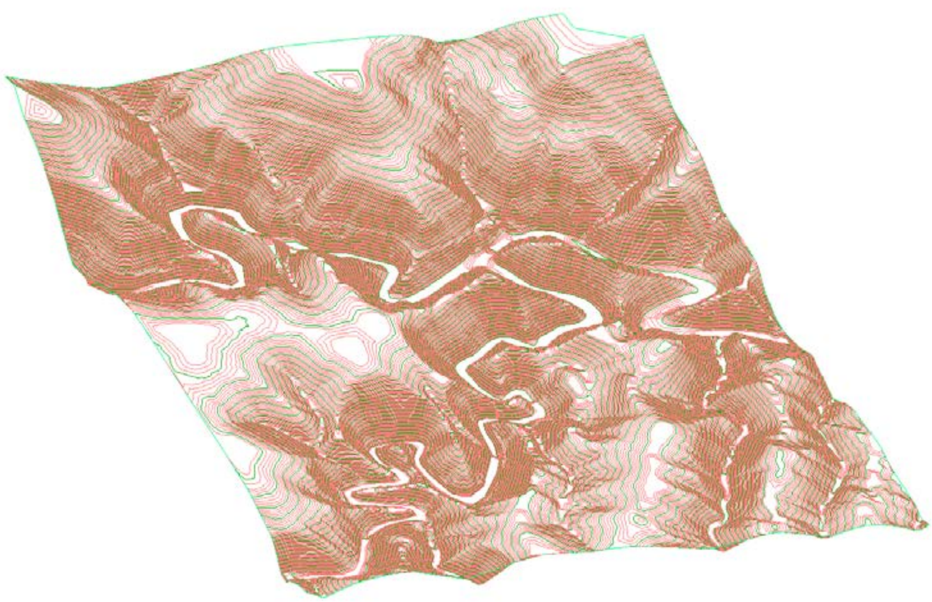


Figure 6 - Isometric view of Civil 3D surface models showing 5-foot contour intervals.

Since 3D models are often developed as part of the design process, project efficiency can be achieved by using the 3D design model to generate the FE model. Additionally, modeling techniques are different enough between programs that modeling in Autodesk products (AutoCAD, Civil 3D, Revit or Inventor) does not lead to an understanding of Truegrid. If the process of drafting (using model development) and developing a FEA model can be combined, a significant time savings can be achieved.

AutoCAD model development is not the focus of this research; however, AutoCAD tutorials were included as attachments for reference:

- Attachment A – AutoCAD Shell Model Development (LOFT)
- Attachment B – AutoCAD Solid Model Development (SWEEP)
- Attachment C – AutoCAD Civil 3D Surface Model Development

LS-PrePost Finite Element Development and Analysis Software

LS-PrePost was developed with full LS-DYNA support in mind. It was designed as a way to visualize LS-DYNA models and edit them, as well as post process models that an analysis has been completed for. LS-PrePost is a pre- and post-processing feature for use with LS-DYNA FE analysis engine. LS-PrePost allows the user to generate FE models and apply all of the required attributes (materials, boundary conditions, loads, etc.) to the model, and will write them to the LS-DYNA input code (also known as key-word input with .k extension).

The Livermore Software Technology Corporation LS-PrePost Online Documentation includes multiple tutorial examples (www.lstc.com/lspp/content/tutorials.shtml) however, they have not been updated to correspond to the new interface. Another resource is "LS-DYNA for Beginners" [6], however that is also based on the old version of the program and does not appear to introduce many topics useful to modeling more advanced structures. It appears that limited resources are available to assist in developing models in LS-PrePost. Since the structural analysis group utilizes LS-DYNA to run FEA on dams, LS-PrePost was selected for this research to determine if it could be used to develop FE dam models based on AutoCAD models developed for design.

Finite Element Model Development

Research Model Overview

LS-PrePost Modeling

LS-PrePost version 4 introduced a secondary GUI that expands on the model editing options. Pressing "F11" will switch between the old and new GUI. The updated GUI is shown in Figure 7 below.

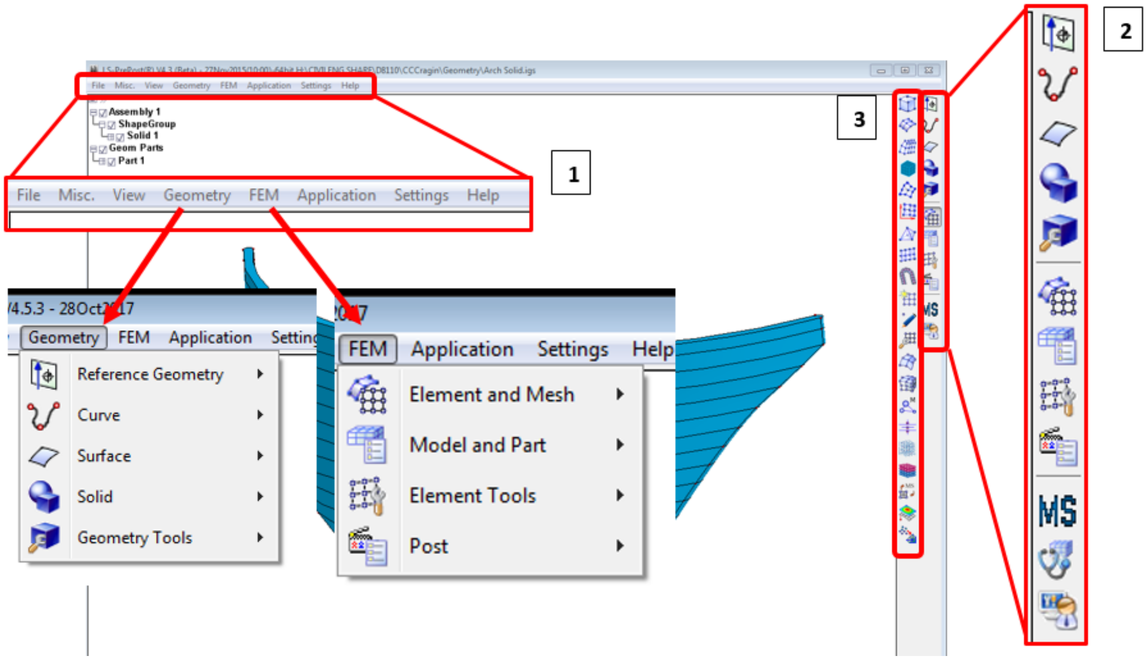
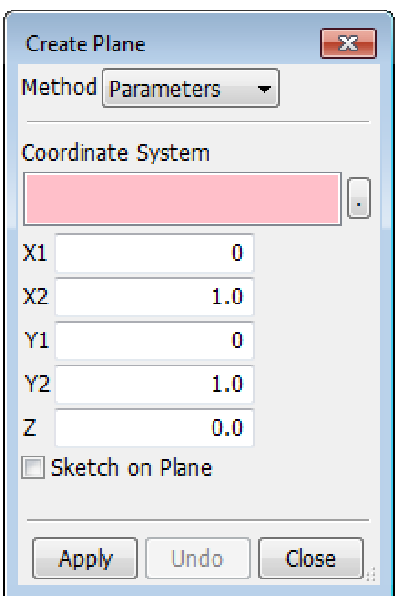


Figure 7 - LE-PrePost Version 4 GUI. Press F11 to switch between the old and new GUIs.

In Figure 7, the toolbar on the top (1) correlates directly to the sidebar options on the far right (2), as shown. The first five options on the sidebar are model geometry options, also found under the “Geometry” tab. The next four sidebar options are FEA meshing and post processing options, found under the “FEA” tab. The last two sidebar options are additional post processing options and a user defined toolbar. Clicking on any of the above options adds additional sub-options on the left side of the sidebar (3) that are also available in the drop-down menus.



Generating Shapes

Similar to CAD programs, the user can generate shapes to convert into FE objects within LS-PrePost if desired. The first five options in the side bar (Figure 7) are dedicated to creating and modifying primitive shapes in LS-PrePost. Most of the shape generation options open a prompt window and the user will need to type in coordinates, while the shape is being generated in the GUI. Figure 8 shows the "Create Plane" option with the preview in the background.

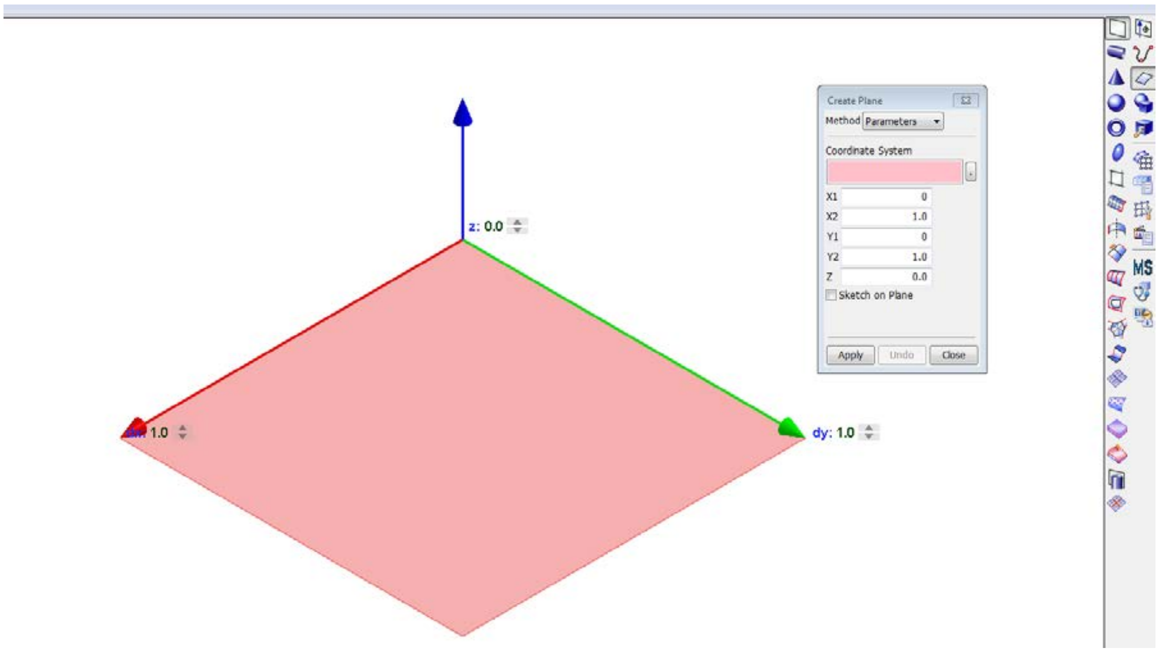


Figure 8 - Creating planes in LS-PrePost.

Generating primitive shapes in LS-PrePost is easy because they are prebuilt in the program. Similar to creating a plane, the prebuilt shapes will preview in the background with the inputs (Figure 9). The prebuilt solids that can be generated are:

- Boxes
- Cylinders
- Cones
- Spheres
- Torus
- Ellipsoids

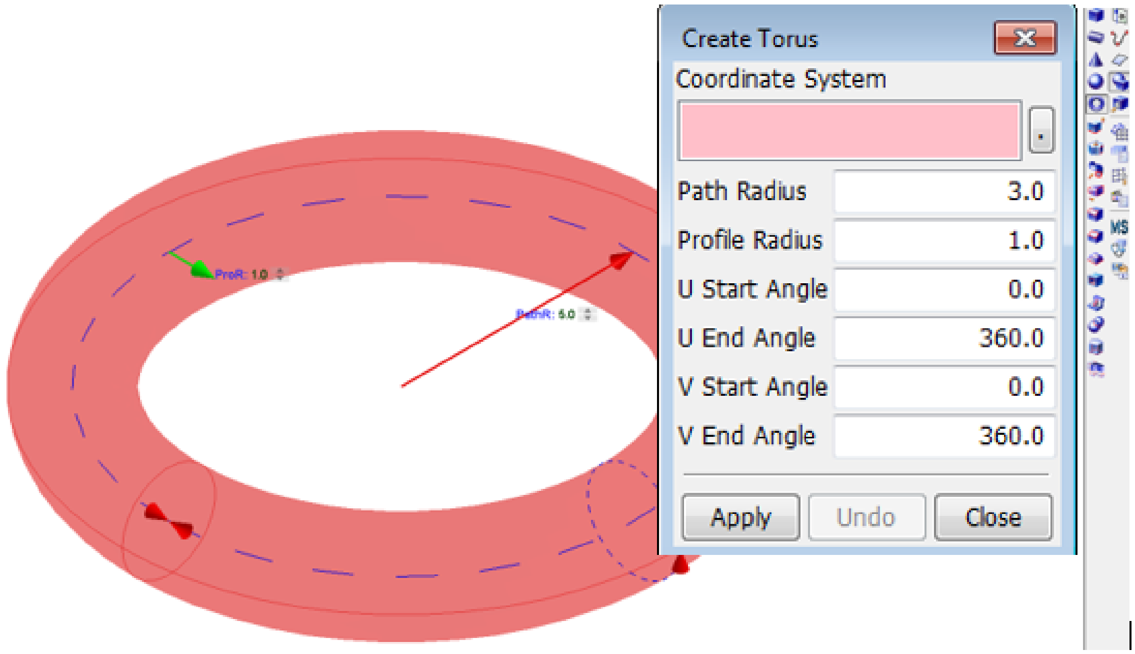


Figure 9 - Generating a torus in LS-PrePost using the built-in shape options.

Additionally, curves may be generated in LS-PrePost to create custom cross sections. These cross sections can be converted to surfaces, which then can be converted to solids through extrusions, lofts or sweeps, similar to the options in AutoCAD.

The geometry generation buttons on the right-hand tool bar in descending order, starting with the second option, is the "Curve" menu, "Surface" menu, and "Solid" menu.

Selecting Model Features

Selecting the desired nodes, elements or shapes can be difficult if the user is not familiar with the interface of the selection input box. The selection input box is organized to specify the following:

1. Selection tool
2. Selection grouping
3. What to do with the selection
4. Filters.

Figure 10 shows the selection tool pop-up. The selection tool section allows the user to choose how the selection is made; e.g. by picking elements or drawing a box around them. Additionally, there is a choice to either add or remove elements from the selection. The selection grouping options specify how selections are made; by elements, by parts, by Sets, etc. The third box in Figure 10 highlights options that the user can select after the desired elements are selected. If the user is going to select the same features again, they can "save" that selection to easily repeat the process. The most powerful options in the selection tool are the filters, highlighted in box 4. The "3DSurf" option ensures that only the outside nodes or elements are selected, making selections significantly easier by not grouping undesired internal nodes. The "Prop" option is the other important selection filter. When the “Prop” option is selected, the “Ang” option needs to be specified. The “Prop” option means that the selection will propagate to other features that are within the specified angle (“Ang”). An entire face of nodes or elements may be selected at once by using the “Prop” function with an appropriate angle. In combination with the “3DSurf” filter, this ensures that only the desired face features will be selected.

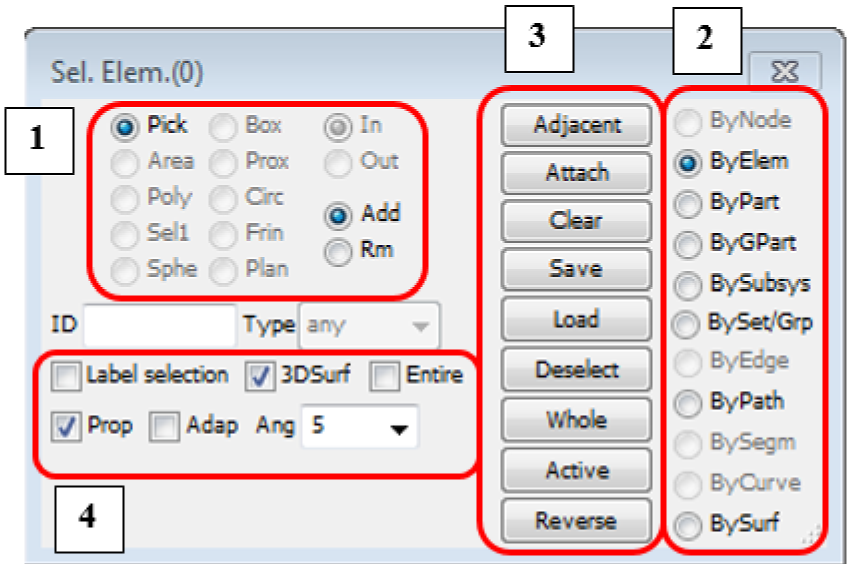


Figure 10 - LS-PrePost selection tool.

Modifying Shape Geometry

Sometimes it will be necessary to modify the geometry of shapes to assist in meshing. The options for modifying shape geometry are included below the “Solid Generation” on the right, or as the last option in the “Geometry” drop down list. Within the modification options, the user can delete or blank elements, extend curves or surfaces, generate objects through offsets, trim shapes, or translate or rotate shapes.

Meshing is easiest for rectangular or near rectangular shapes when using hexahedral elements, which are typically preferred to tetrahedral elements due to model size limitations. To assist in meshing, the shapes may be divided into smaller, more workable sections using reference planes and trim commands. Below is a discussion on trimming shapes to simplify the meshing operations, as well as highlight some of the transform and trimming options.

To divide the shapes into smaller, more basic shapes, start by inserting reference planes where the model is to be divided into sections. Reference planes may be added from the “Reference Geometry” toolbar by selecting the “Reference Planes” icon or drop down option. There are many options to add reference planes; specified coordinates may be typed, 3 points may be selected, or faces of objects may be selected, among other options. For this model, the “3 Points” method was used.

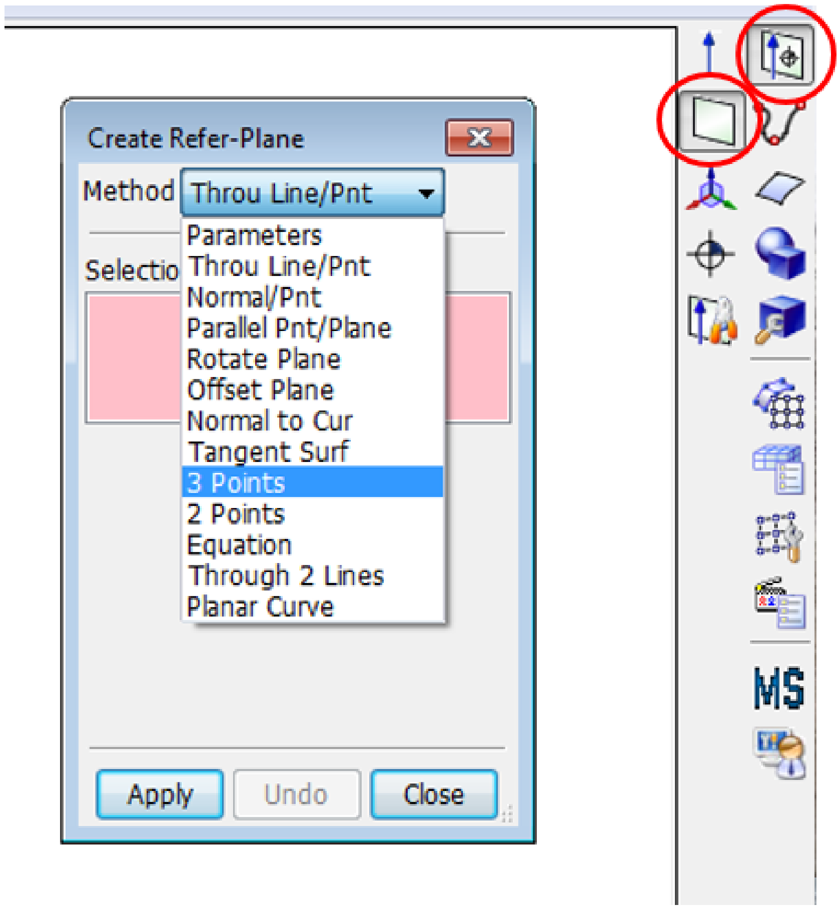


Figure 11 - LS-PrePost "Create Refer-Plane" menu.

The reference planes may be moved if a different location is desired. Moving the reference plane, or any shape, can be performed using the “Geometry Tools” “Transform” command. The “Transform” menu is shown in Figure 12.

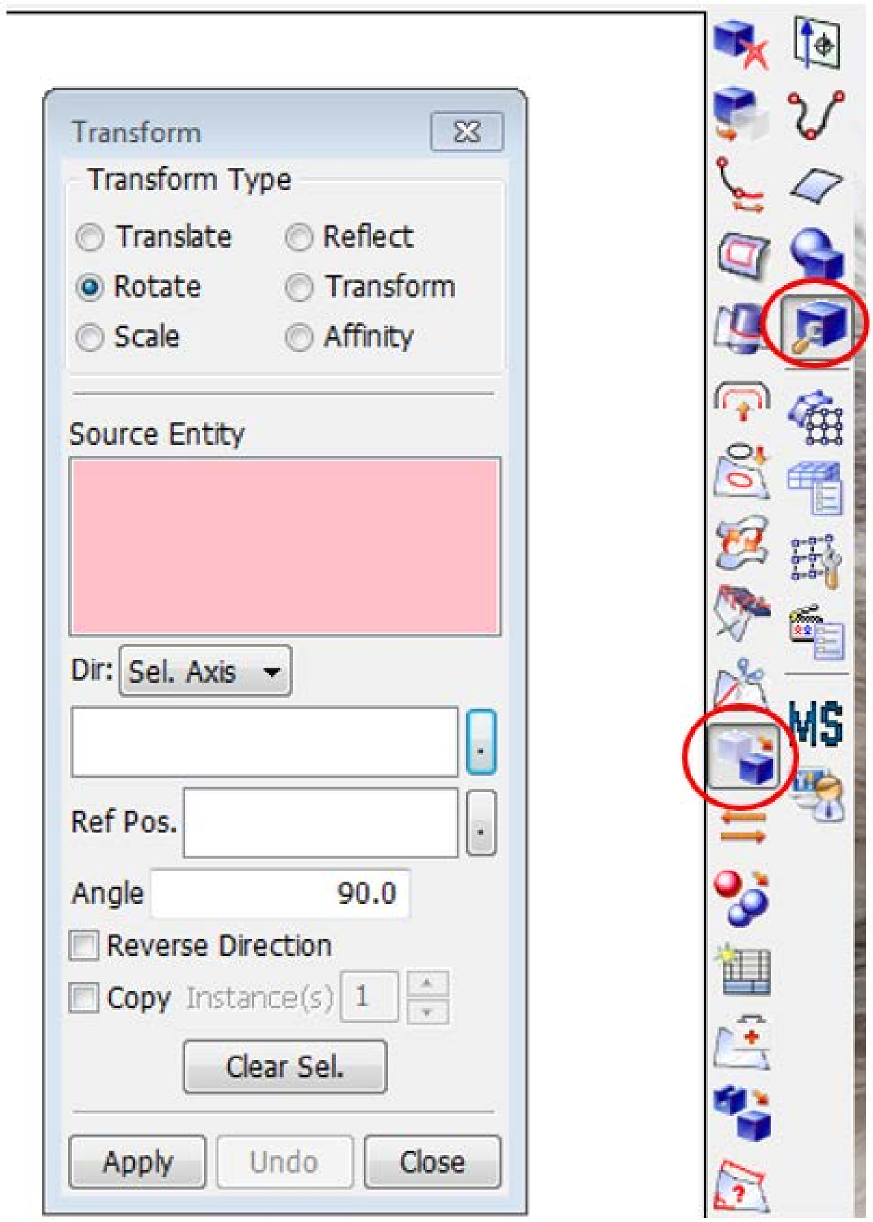


Figure 12 - LS-PrePost "Transform" menu.

Once the reference planes are located where the shapes are to be divided, they are ready to be trimmed. In the geometry editor, open the “Trim” option, then “Cut Solid” to create multiple solids around a specified cutting edge (the reference plane in this case). Figure 13 shows the model with reference planes, Figure 14 shows the process of trimming a solid into multiple parts.

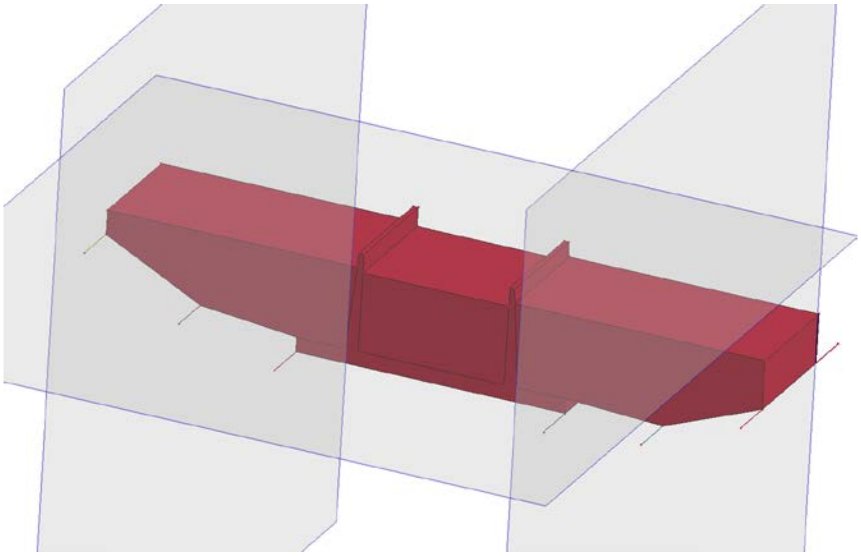


Figure 13 - LS-PrePost reference planes to cut model into smaller, more basic shapes.

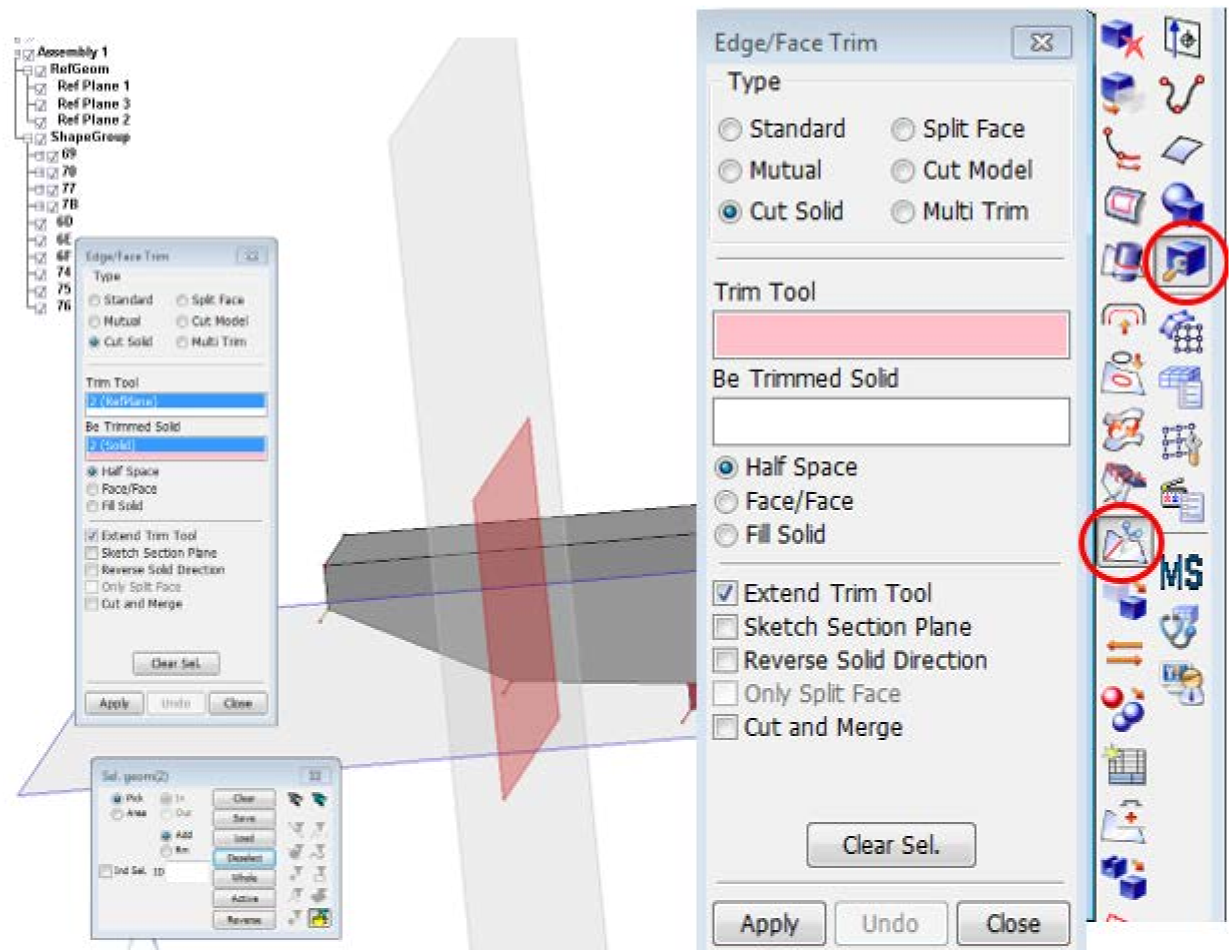


Figure 14 - LS-PrePost trim solid options. The menu on the left is blown up on the right side of the figure.

When the model is divided as desired for FE modeling, the user is ready to begin generating the elements.

Finite Element Modeling

When the model is ready for creating the FE objects, select the "Element and Mesh" menu option to open the meshing options. Depending on the problem, either shell or solid meshes may be developed, with both options found in the "Element and Mesh" menu. For shell modeling, a surface can be generated using the following tools discussed in this report:

- Auto Mesher
- N-Line Mesher
- Element Generation

Similarly, there are multiple ways to generate solid meshes in LS-PrePost. The tools that were utilized in this study were the:

- Solid Mesher
- Tetrahedron Mesher
- Element Generation
- Block Mesher

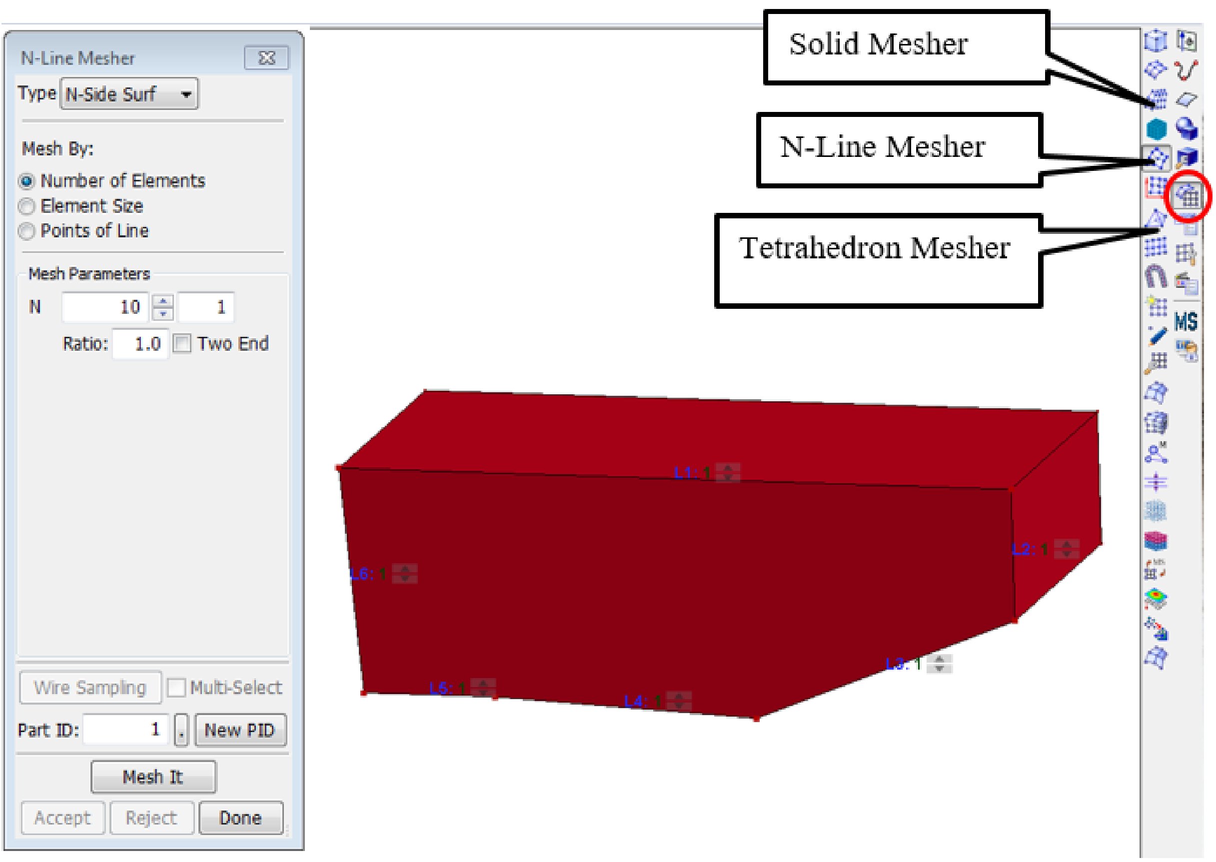


Figure 15 - Select solid meshing options from the "Element and Mesh" menu.

Multiple options are available to create the FE models for the FEA, besides the options discussed in this report. Additionally, all of these techniques will likely be combined when generating a complete model. The completed Basic Model is shown in Figure 16, was created using a combination of the "Solid Mesher" and "N-Line Mesher" commands. The techniques highlighted above are demonstrated below in the element generation for either the Basic Model or the Advanced Model in this report.

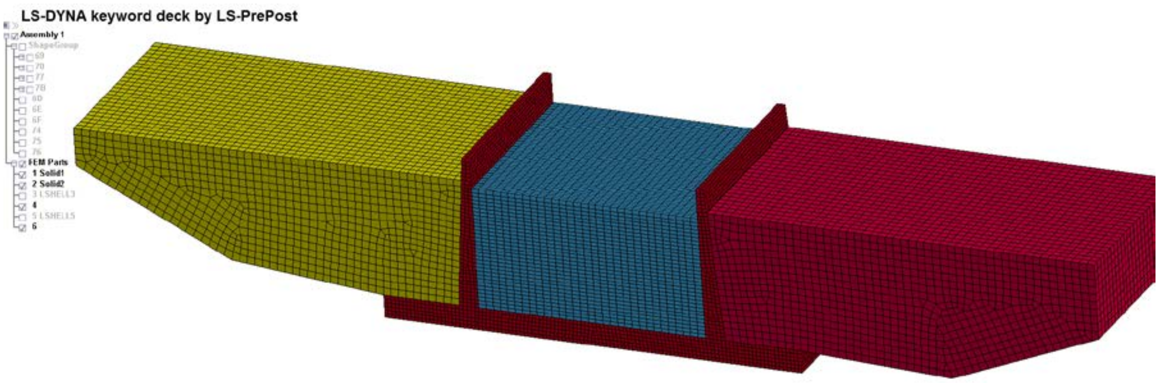


Figure 16 - Basic Model meshed using the Solid Mesher and N-Line Mesher for FEA.

Development of the Basic Model

A basic model of a typical spillway stilling basin structure was completed using AutoCAD and LS-PrePost. The structure consists of a spillway stilling basin with cantilever walls, water contained in the spillway, and soil behind the spillway walls. Figure 17 shows the model in AutoCAD.

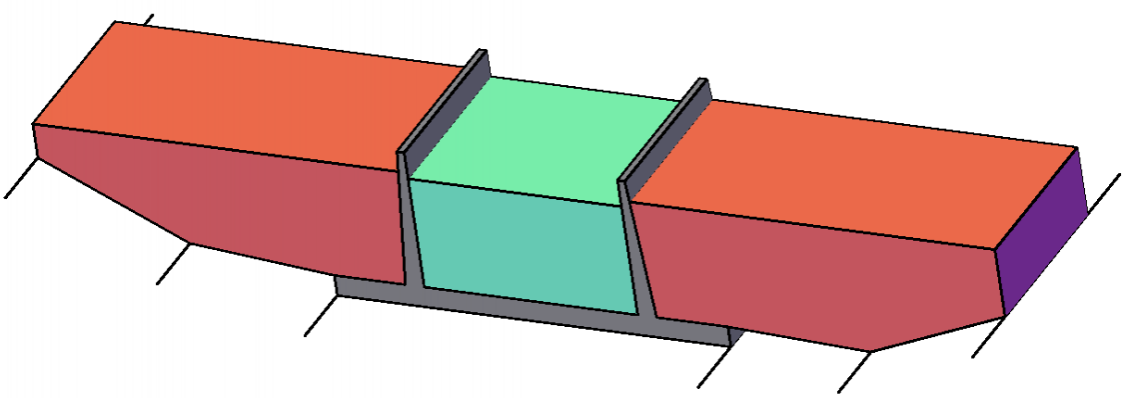


Figure 17 - AutoCAD rendition of Basic Model for FEA modeling research.

The Basic Model was designed in AutoCAD as a 20-foot tall cantilever walled spillway chute, with a depth of 25 feet. The model, as shown in Figure 17, was imported into LS-PrePost to develop the FEA model. To import the model, the AutoCAD model was exported as an IGES, then imported into LS-PrePost as an IGES. For details, see the AutoCAD tutorial Attachment A.

The LS-PrePost model is shown in Figure 18. As can be seen in the figure, each shape is brought into LS-PrePost as a separate shape (also known as "parts" in FE model). Although the objects are rendered in LS-PrePost, these shape objects are not FE objects and cannot be directly used for modeling.

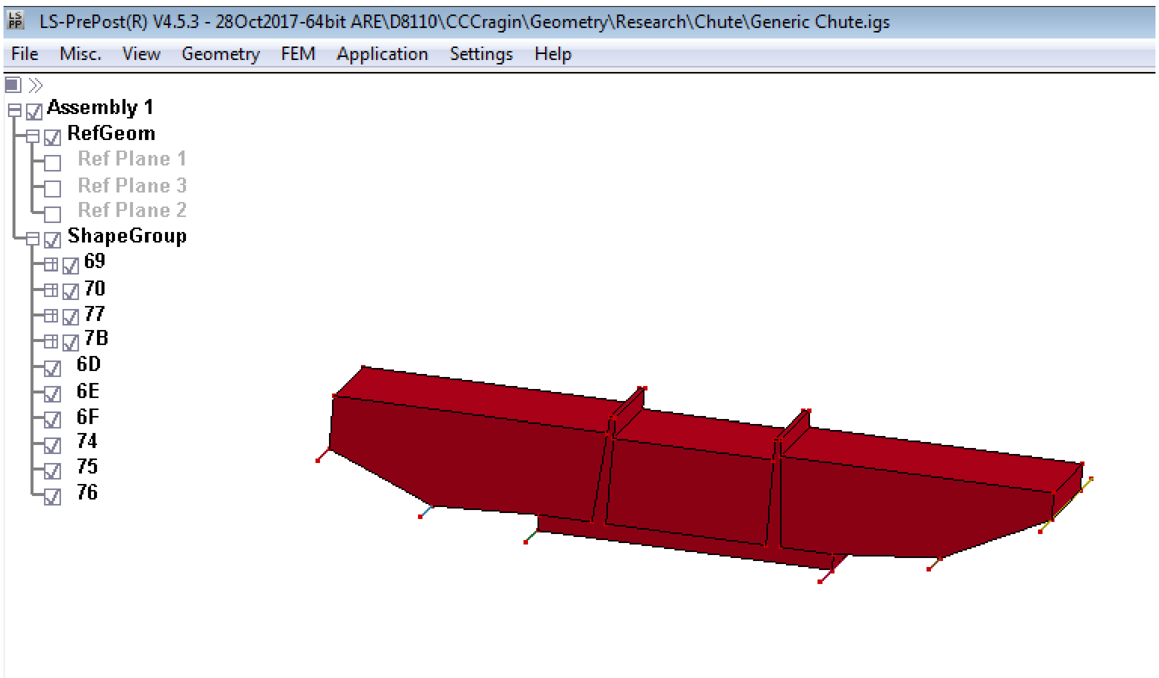


Figure 18 - Basic model shape objects in LS-PrePost after importing IGES elements.

After importing the .iges file shapes, the user may begin meshing as is, or, the user may modify the solids into more generic elements for a more generalized mesh later. Both options will be shown below. The specific option selected should be based on the geometry that is imported. It is possible to generate shapes within LS-PrePost, however, it is much easier to generate the shapes or outline line work in a CAD program and import into LS-PrePost.

Solid Mesher

The "Solid Mesher" will automatically mesh a solid shape if the program can find a closed volume object, such as one of the .iges shapes imported from AutoCAD. Following selecting the "Solid Mesher" tool, select "Try Meshing Automatically" after selecting the desired solid. If LS-PrePost can find a closed volume it will suggest a mesh; the user must "Accept" or "Reject" the mesh. Note: If the user selects "Done" prior to accepting the mesh, the mesh will be rejected, this is typical for all meshing operations, or more generally, most operations within LS-PrePost.

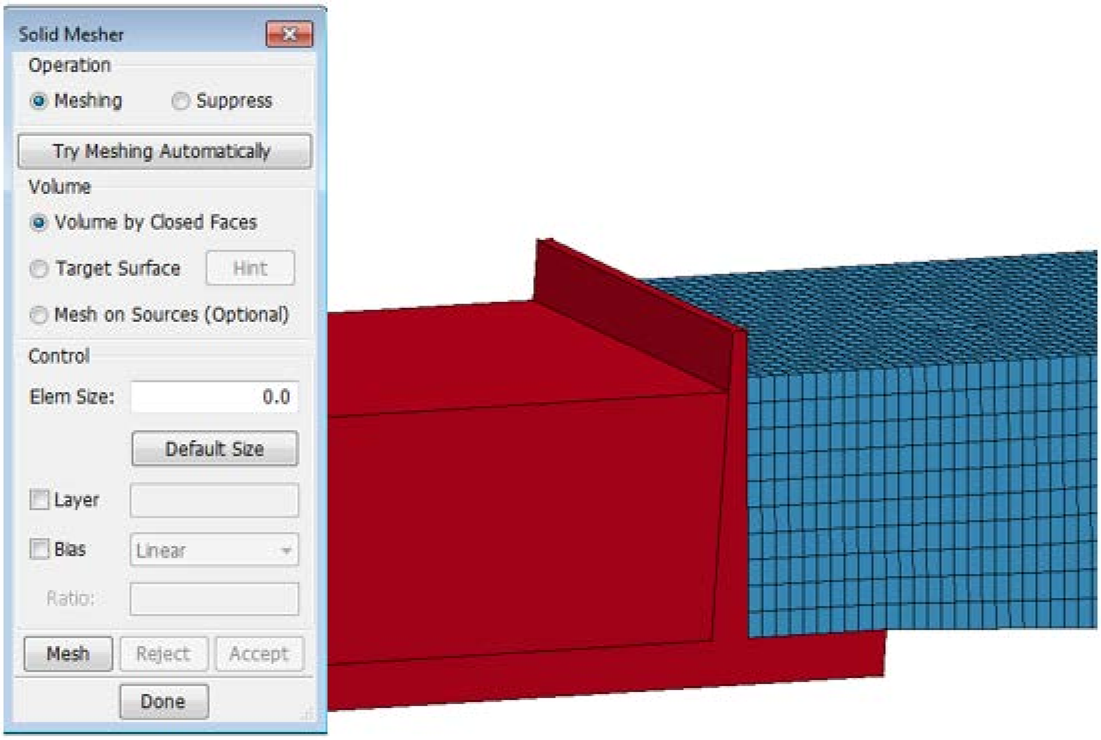


Figure 19 - Solid Mesher automatic meshing of the soil on the right of the spillway.

The biggest advantage of the "Solid Mesher" is the speed of meshing and the use of hexahedral elements. The average element size can be changed; however, the program dictates the mesh layout. While the "Solid Mesher" is fast, it only works well with basic shapes, complex shapes may cause LS-PrePost to crash. The exact cause of crashing is unknown; however, it is presumed to be correlated to computing power because crashing the program could be replicated when trying to mesh the same shape. As a precaution, prior to using the "Solid Mesher" be sure to save the program.

Tetrahedral Mesher

Similar to the "Solid Mesher", the Tetrahedron Mesher will automatically create a 3D mesh for a solid object. However, instead of hexahedral elements, the "Tetrahedron Mesher" utilizes tetrahedron (4 or 10 node) elements. This allows for automatic meshing of more complex shapes; however, there are significantly more elements when using a tetrahedral element. It is recommended that if tetrahedral elements are used that the 10-node option (quadratic) is selected due to the high stiffness observed using 4 node tetrahedral elements [7].

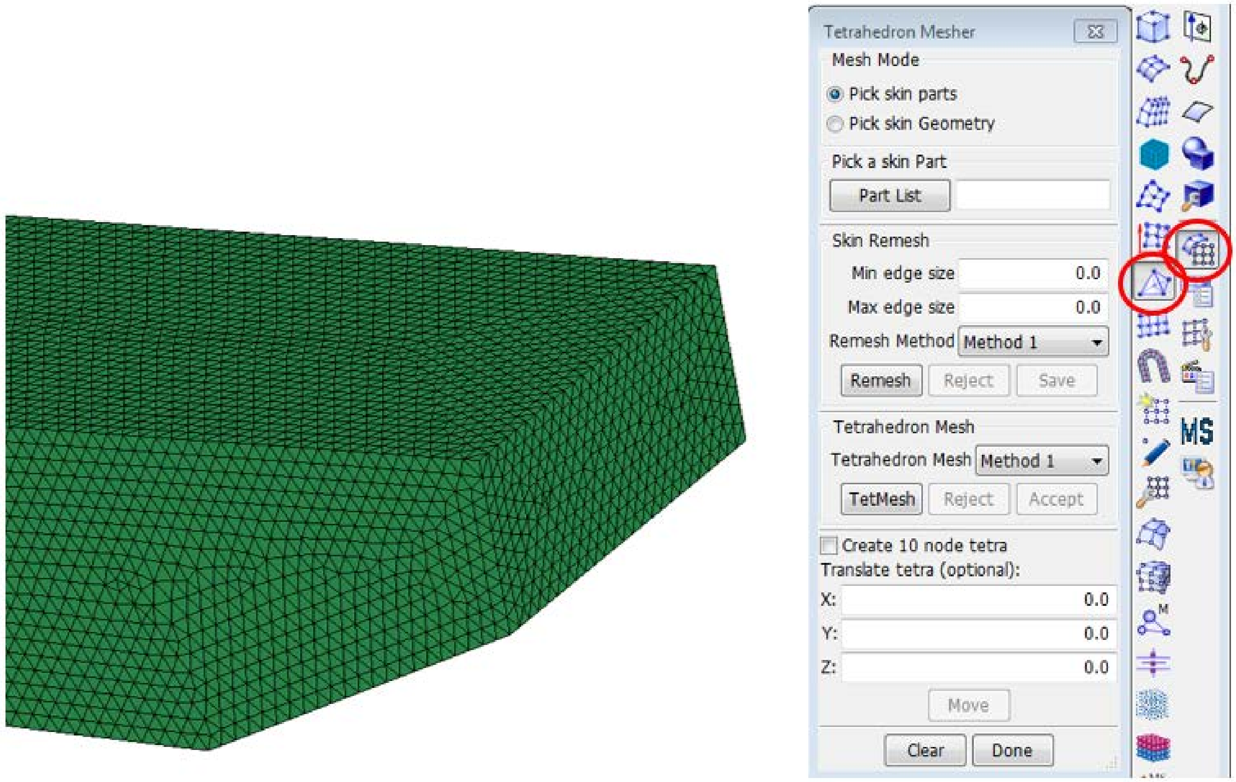


Figure 20 - Meshing the model using the Tetrahedron Mesher.

Creating a tetrahedral mesh is a simple two-step process: In the "Mesh Mode," at the top of the "Tetrahedron Mesher" popup, select "Pick Skin Geometry." In the menu, parts refer to FE meshes while geometry refers to non FE shapes. Then, select the faces of the solid object, specify the edge size (controlling the element size), and select "Trial Mesh." The trial mesh is the first step before making the tetrahedral elements. The trial mesh is a preview of the FEA object. If the mesh appears satisfactory, select the "Create 10 node tetra" option, the select "TetMesh" to create the tetrahedral elements. "Accept" the mesh if it is acceptable. Figure 20 shows an example of meshing using the "Tetrahedron Mesher".

The advantage of using the "Tetrahedron Mesher" is how quickly solid FE meshes may be generated and how accurately they can represent the actual shape of the object being modeled. However, the drawback to using the "Tetrahedron Mesher" is the model size. As mentioned above, the tetrahedron mesh will create significantly more elements and nodes, especially when using the recommended 10-node elements. Table 1 shows the model summary for the parts developed above using the "Solid Mesher" and the "Tetrahedron Mesher", based on approximately 1-foot edge element sizes:

Table 1 - Tetrahedral versus hexahedral mesh element comparison.

Mesh Type	Nodes	Solids
Tetrahedral (10 Node)	112826	76868
Hexahedral	14913	12680

N-Line Mesher

Using the “N-Line Mesher” to generate a solid mesh is more similar to a semi-automatic mesh operation than the automatic mesh operation of the “Solid” or “Tetrahedron Meshers”. The “N-Line Mesher” takes more steps, but the ability to customize the meshing operation may be preferable for some parts in an FE model. The “N-Line Mesher” will be used to create a shell mesh of triangular or quadratic elements that can be extruded to a 3D solid composed of mostly hexahedral elements.

Start by opening the “N-Line Mesher” window and selecting “N-Side Surf.” This allows the user to define the surface that will be meshed by selecting the boundary lines. On each line that is selected, “Ln:x” will appear, where “n” denotes the line number and “x” the number of nodes which that line will be broken into for mesh development (Figure 21).

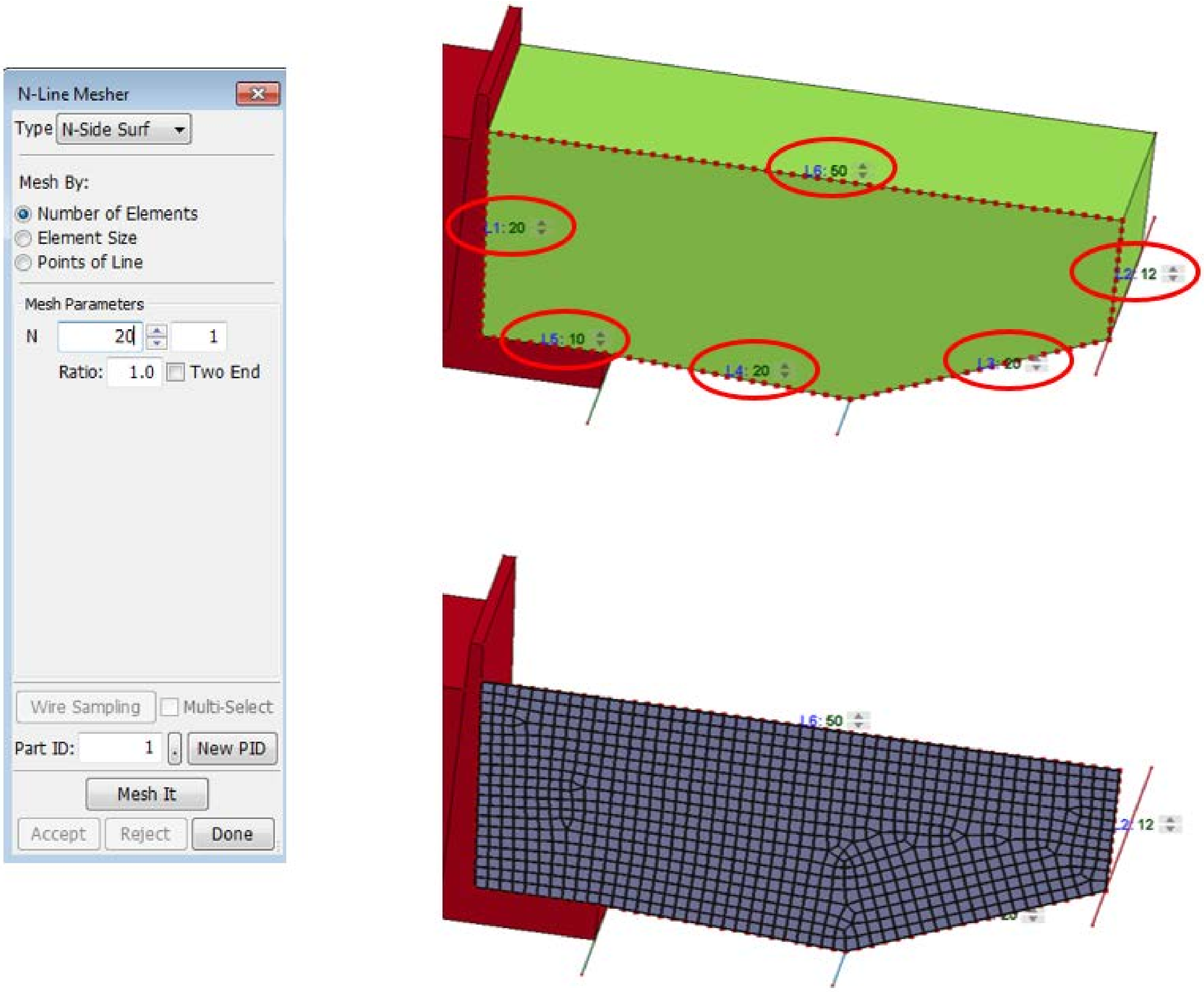


Figure 21 - N-Line Mesher setup on 3D shape surface and completed 2D Shell Mesh

For meshing, based on the proposed layout, select the “Mesh It” button. If the mesh is acceptable, select “Accept”, otherwise the user may change the nodes per line and re-mesh until the desired mesh is created. Notice in Figure 21 that both triangular and quadrilateral shell elements were used to develop the mesh. To use only quadrilateral elements, the “Auto Mesher” can be used to re-mesh the 2D part into a specific mesh type (Figure 22). The “Auto Mesher” is the automatic meshing tool to generate or modify shell element meshes.

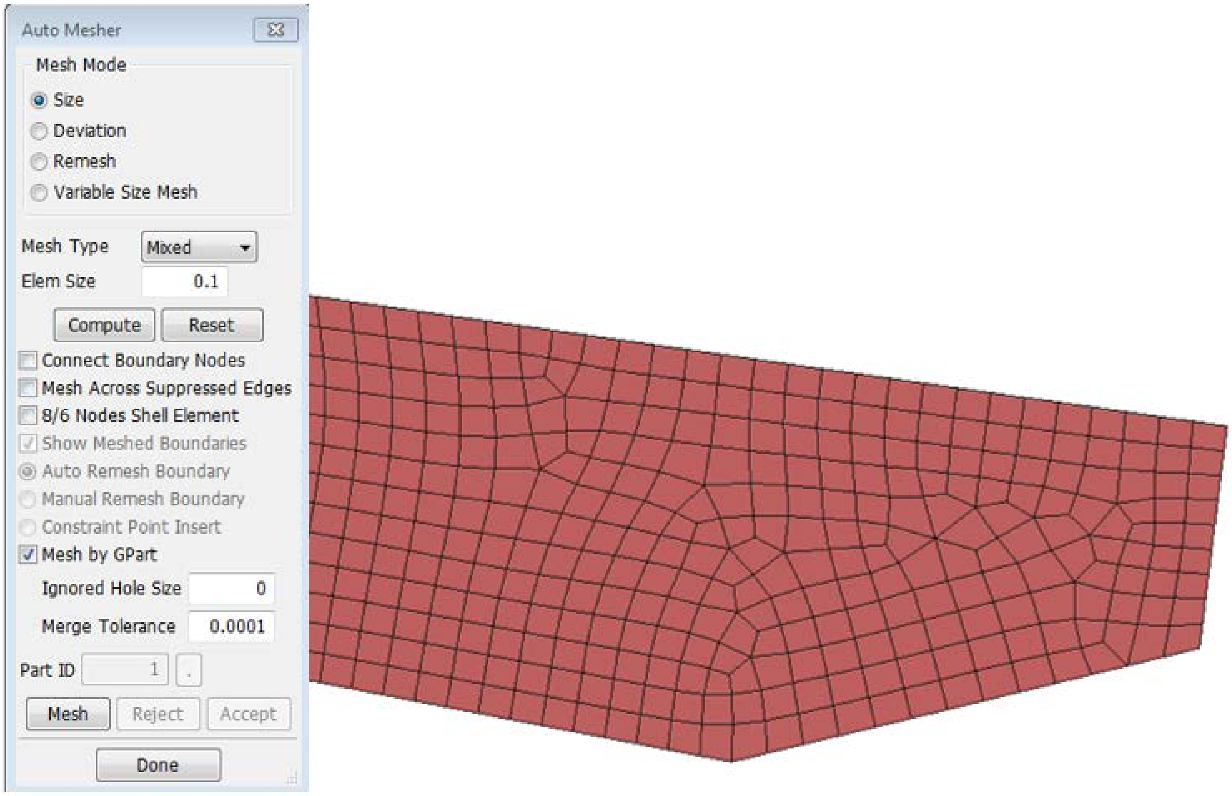


Figure 22 – Re-Meshing the N-Line Mesh into only quadrilateral elements.

When the desired 2D mesh has been developed, the mesh needs to be extruded into a 3D mesh with solid elements. To extrude the 2D mesh into solid (3D) elements, open the "Element Generation" menu. Make sure the "Solid" option is selected, then change the "Solid By:" option to "Shell Sweep" to sweep the 2D shells along a path. In this case the path is one of the 3D shape object edges. When specifying the "Sweep Path", set the number of segments; this is the number of solid elements that will be created along the sweep line, similar to the number of nodes during "N-Line Meshing." Select "Create" then "Accept" (Figure 23) to generate and produce the 3D FE Part (Figure 24).

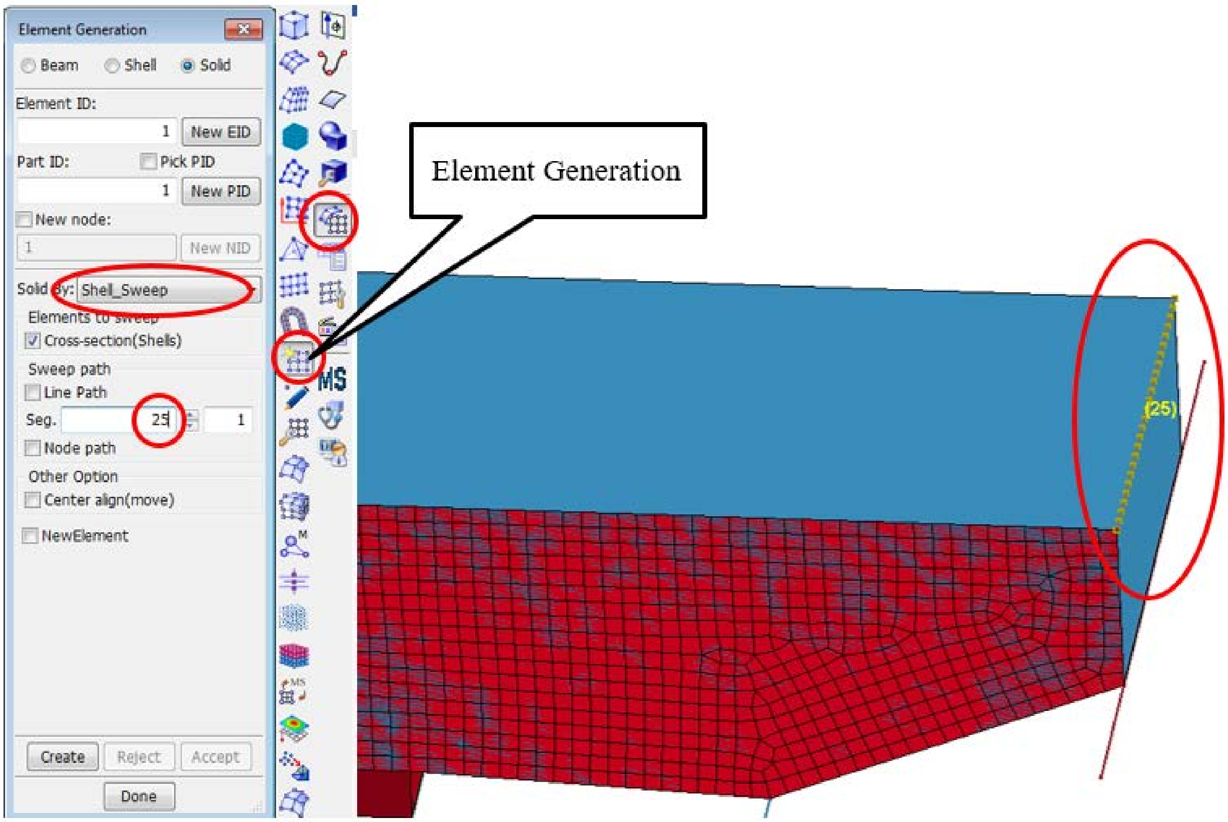


Figure 23 - Element Generation menu to extrude surface elements into solid elements.

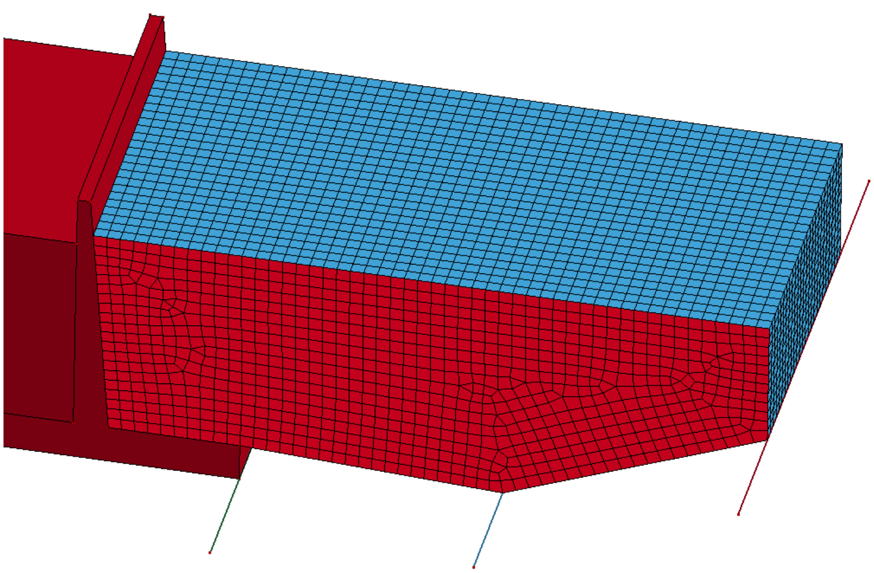


Figure 24 - 3D Solid elements from shell sweep command.

General Mesh Editing

As seen above, mesh generation is fairly automatic in LS-PrePost, however, there are ways to edit the mesh so it is acceptable for the problem that is being solved. Within the element editing tools, the modeler may split elements, merge elements, or check the elements to see if there may be errors due to modeling.

A typical modification that is made to a model is to have a finer mesh along a more critical area of the model. An example of this is shown in Figure 25. It is advised to modify the 2D shell part prior to extruding to a solid mesh so the “Retain Bdy” (Retain Boundary) condition may be specified; otherwise, the solid editor will split elements without maintaining nodal contact from one element to the next. Additionally, the shell editing options allow for significantly more complex operations than the solid mesh editing options. The 2D shell part in Figure 25 was split below with the retain boundary option applied. If desired, the mesh may be re-meshed using the “Auto Mesher” (Figure 22) to retain quadrilateral elements, or the “Element Editing” menu can be used to manually create quadrilateral shell elements under the “Create” options.

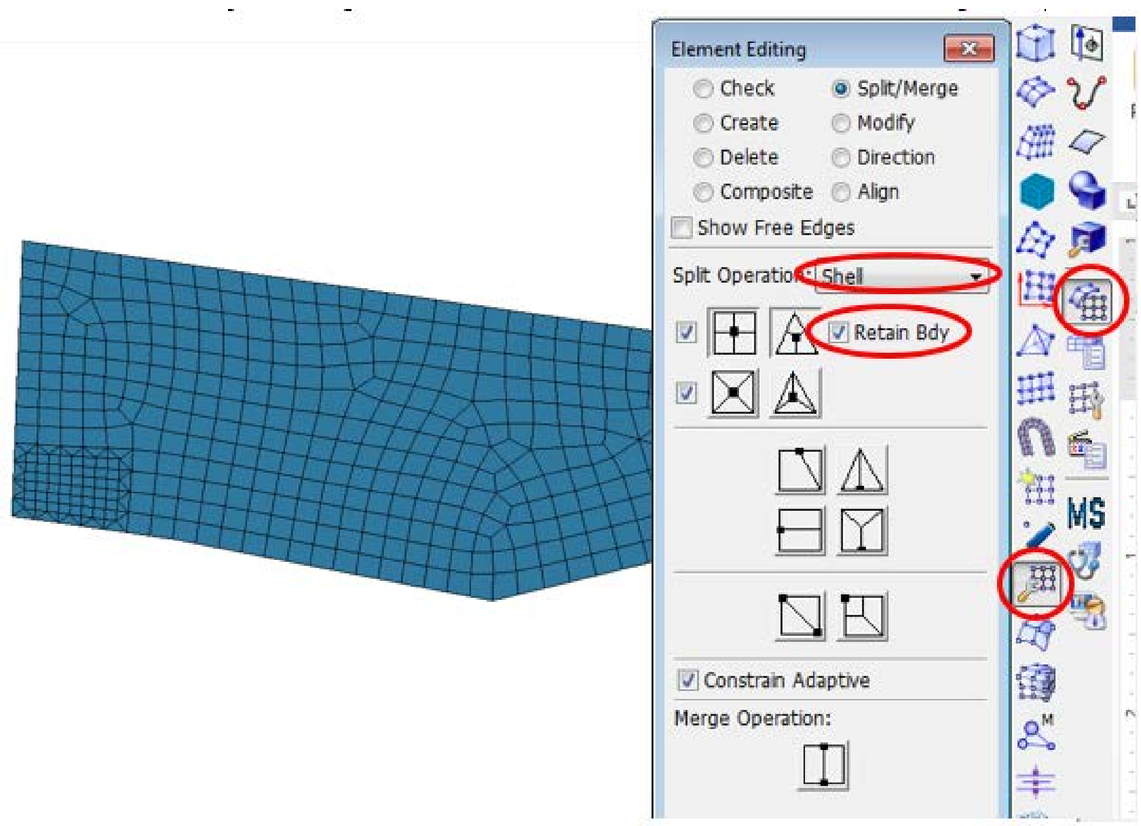


Figure 25 – LS-PrePost Element editing menu to refine a mesh in a critical area.

Assigning Properties

To solve FE problems more than just the geometry needs to be constructed. FE models take into consideration boundary condition, material properties, and how different materials interact, as well as external features such as applied loadings. To complete the FE model these features will need to be added to the model. Additionally, LS-DYNA modeling operations need to be added as well.

To define the required model properties, the “Keyword Manager” and “Entity Creation Manager” will be used within the “Model and Part” submenu in the FEM menu. Figure 26 shows the “Model and Part” options where the “Keyword Manager” and “Entity Creation Manager” are accessed.

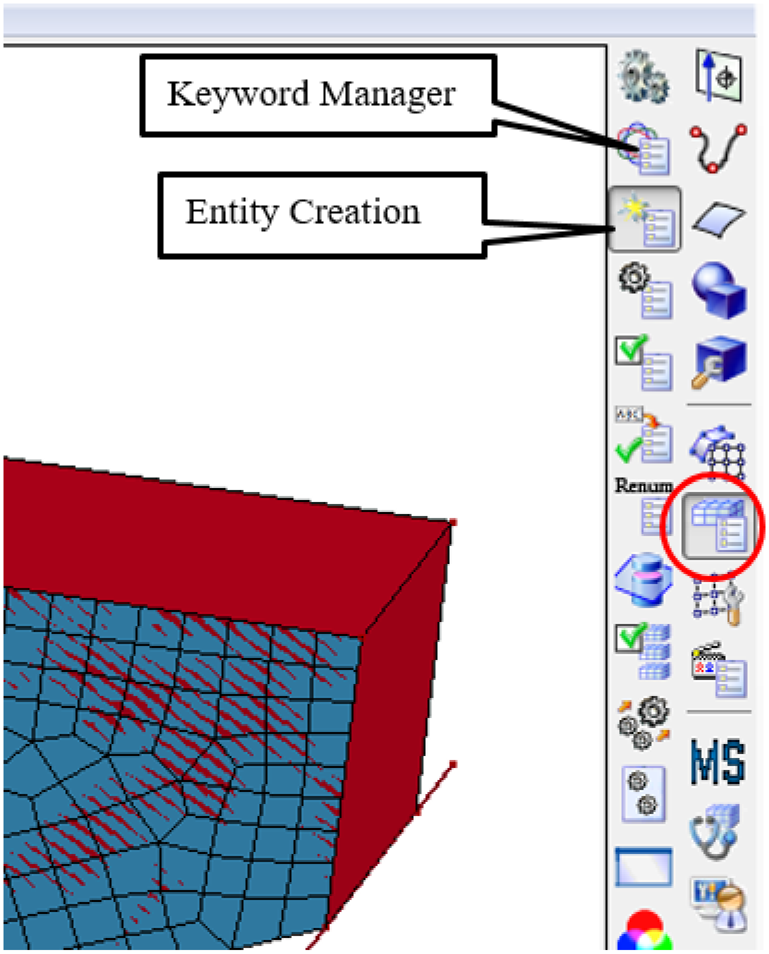


Figure 26 - LS-PrePost Model and Part options.

LS-DYNA defines all the input in a card format. To add in the material property, boundary condition, contact, or any other required definitions, cards will be used. The cards are organized by keywords, and the keywords are defined in the “Keyword Manager.” Typically, there is no specific order that these features need to be applied when building the model; the program will write the LS-DYNA analysis keyword model file (.k file) in the appropriate order. After defining the geometry, the “Keyword Manager” will be the most utilized option in LS-PrePost. An example using “Keyword Manager” is included below.

In addition to the “Keyword Manager,” the “Entity Creation” manager will be used to define “Sets,” or groups of objects that will be assigned similar properties (i.e. loads, contacts, or boundary conditions). Sets are required to assign contacts or boundary conditions to, and thus are required for the model. Defining Sets is discussed in further detail below.

Keyword Manager

Keywords are defined in the "Keyword Manager." The "Keyword Manager" is where the user creates cards to assign to the objects or define the model. Figure 27 shows the "Keyword Manager" menu.

This tutorial addresses the basic keyword cards that are typically used in an LS-DYNA model, however, depending on different features, different keyword cards can be used. For more details, see the LS-DYNA Keyword User's Manual; most of the keywords are defined in the Manual [8], while newer cards are actively being developed for new features.

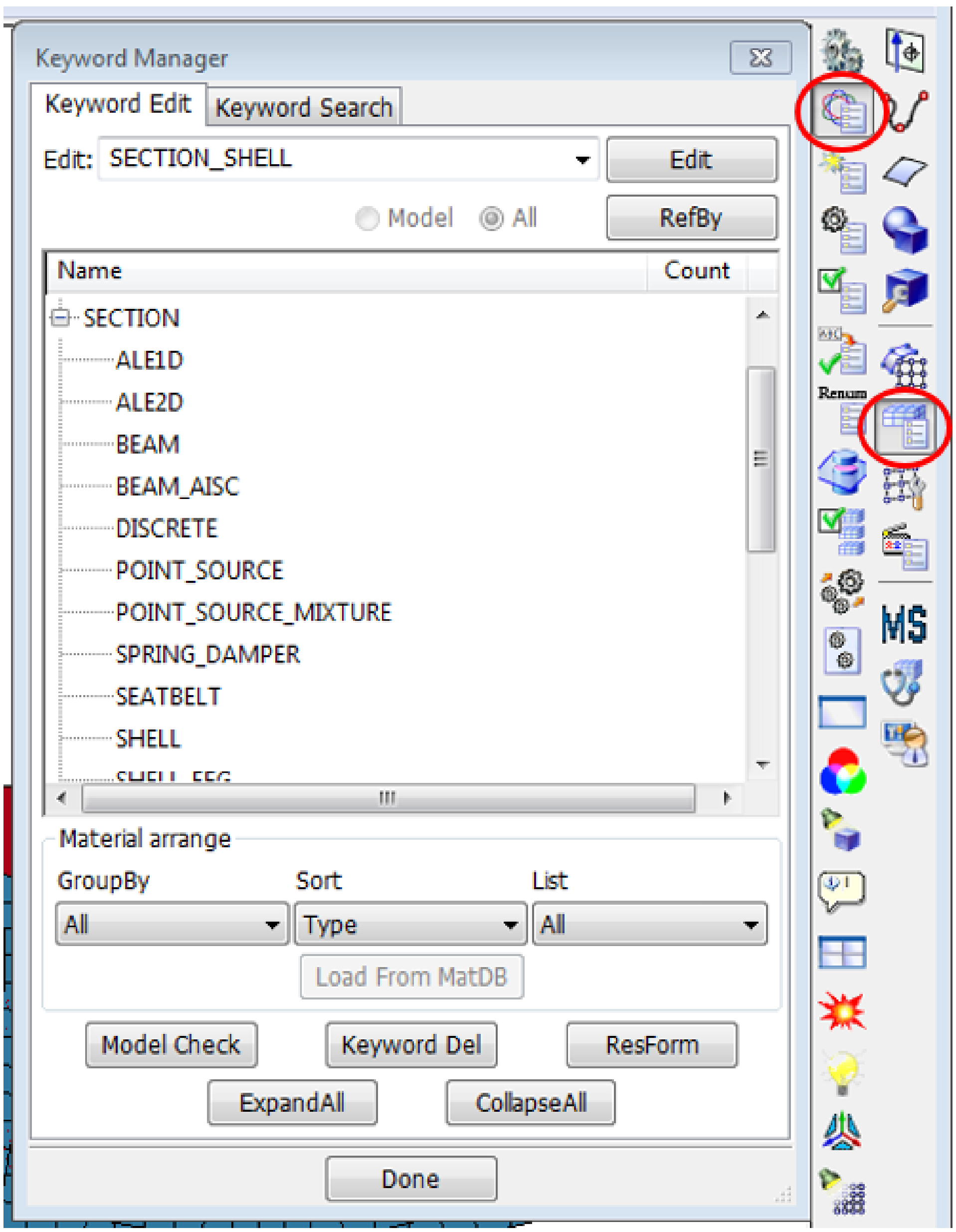


Figure 27 - LS-PrePost Keyword Manager and Keyword Input Form.

Typically, the following keywords should be added to a model:

- Part (Figure 28)
  - This defines the actual part that was modeled and assigns "Section", "Material", and "Hourglass" cards to a part.

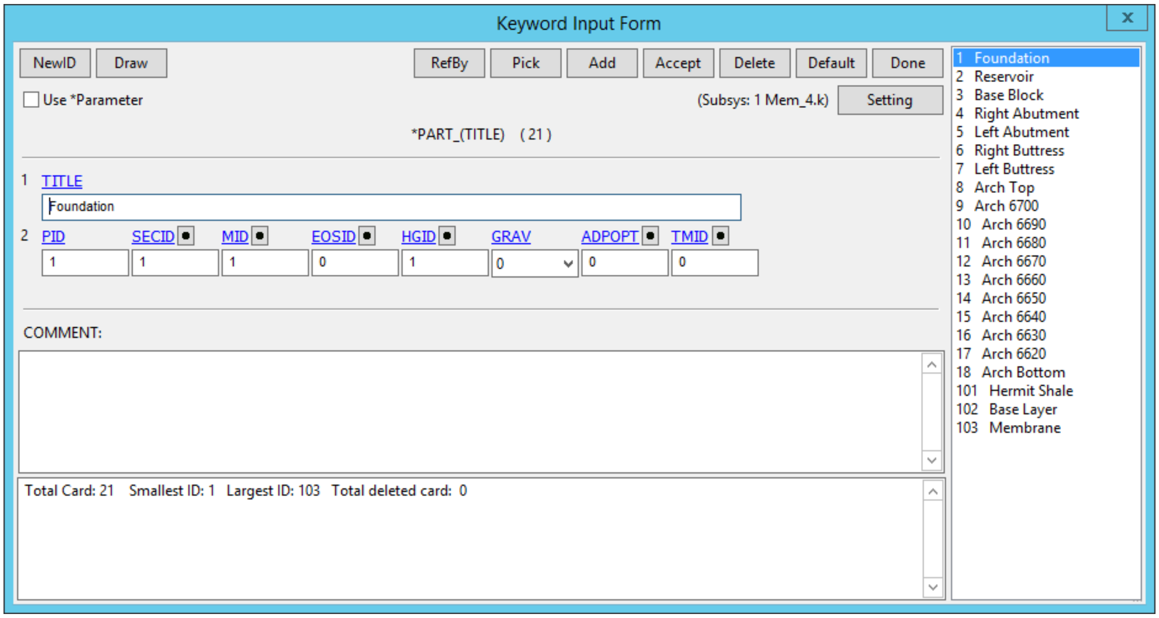


Figure 28 - "Part" card. All cards look similar to this, with more or less options as required.

- Section
  - "Section" cards define the part element type, such as solids, shells, or beams.
- Material
  - "Material" cards define the different material types and the material properties that will be used in the model.
  - Note: many different material models can be used. For more details see the LS-DYNA Keyword User's Manual [8].
- Contacts
  - "Contact" cards define the contact between materials or nodes of different parts, such as a sliding surface or a tied surface. Sets are used to define the specific parts of the model that the contacts apply to.
- Loads
  - Assigning gravity loads or loads in a specific direction. Typically combined with "Define_Curve" cards to define how and when the load is applied.
- Define Curve
  - "Define Curve" can be used to define things that change over time (e.g. loads) or apply a velocity to a model.
- Hourglass
  - "Hourglass" cards define the hourglass and bulk viscosity of objects; this card is referenced in the "Part" card.

- Keyword
  - The "Keyword" card defines the memory allocated toward solving the FEA and the number of computer processors dedicated to solving. Note, each CPU assigned requires 1 license.
- Control Cards
  - "Control" cards can be used to change the default options on other cards, however, the following two are highly recommended.
- Control_Termination
  - Set the End Time to the desired calculation time (also analysis time) and set everything else to "0."
- Control_Timestep
  - This is the solver timestep for the model.
- Boundary
  - "Boundary" cards are used to define the edge of models, similar to contacts between parts. Sets need to be assigned to the boundary to define where it is.

Sets

Sets are used to group features (surfaces, solids, nodes, parts, etc.) together so that similar properties can be assigned. These properties can vary from assigned loads, boundary conditions, or contact surfaces. The "Entity Creation" menu with the Set options expanded is included in Figure 29.

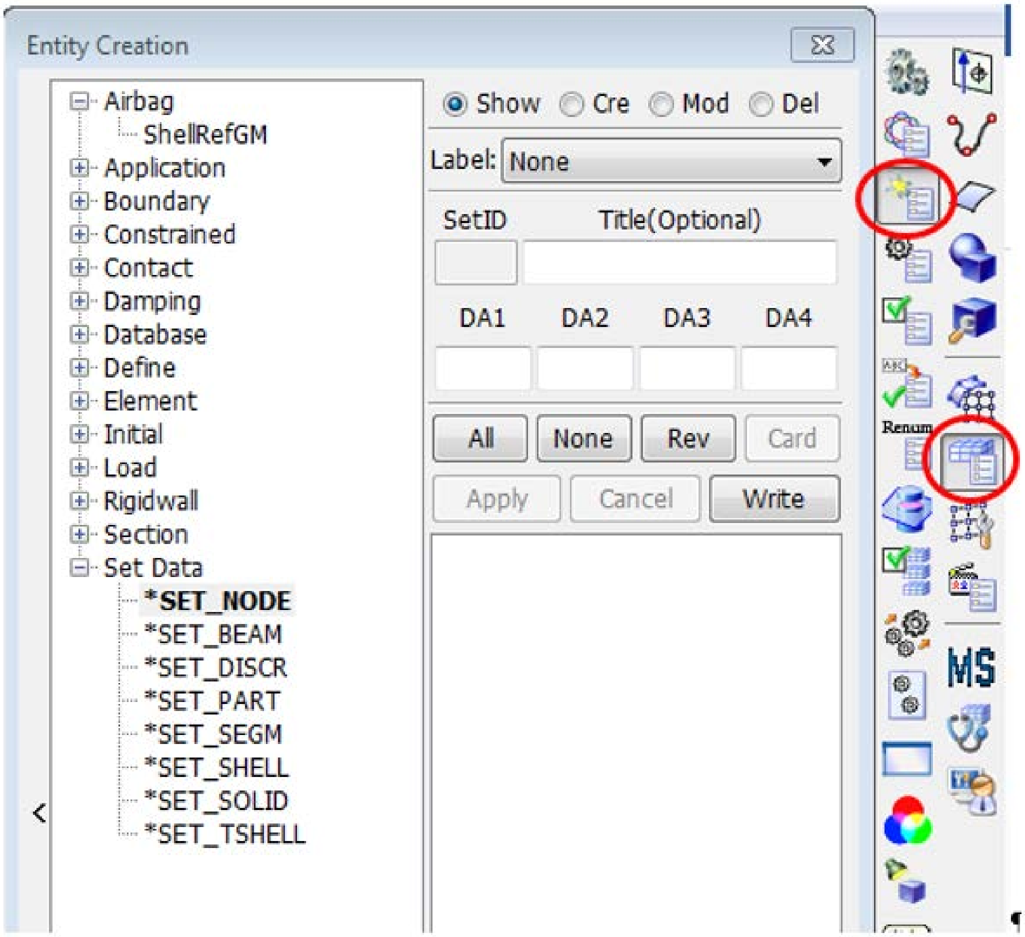


Figure 29 - Entity Creation menu used for assigning Sets.

The most common Sets used in FE models are Node, Part, or Segment Sets. As the title implies, Node Sets are groups of nodes, Part Sets are groups of parts, while Segment Sets are element faces. Specific cards require specific types of Sets. For example, the contact card “Nodes to Surface” requires a Node Set and a Segment Set, while “Tied Surface to Surface” requires two segment Sets. Part Sets are typically used to assign which parts a specific load is applied to, such as gravity, since gravity is typically applied to all parts except the foundation.

For the Basic Model Node and Segment Sets and were used. The first Node Set created is the bottom nodes on the spillway slab. These nodes will be given boundary conditions to represent the foundation. To create the Node Set, select the “Cre” (create) option in the “Entity Creation” menu, then select the desired nodes using the selection filters discussed above. Figure shows the “Entity Creation” menu used for creating Node Sets. To create the Node Set, open the “Entity Creation” menu, select the desired nodes, then “Apply” the *Set_Node card. Do not worry about the DA1 thorough DA4 options as they will autofill in the card based on the selection. Figure 30 shows the application of the menu. Note: the left-hand menu that is not clearly visible is the same menu shown in Figure 29, above.

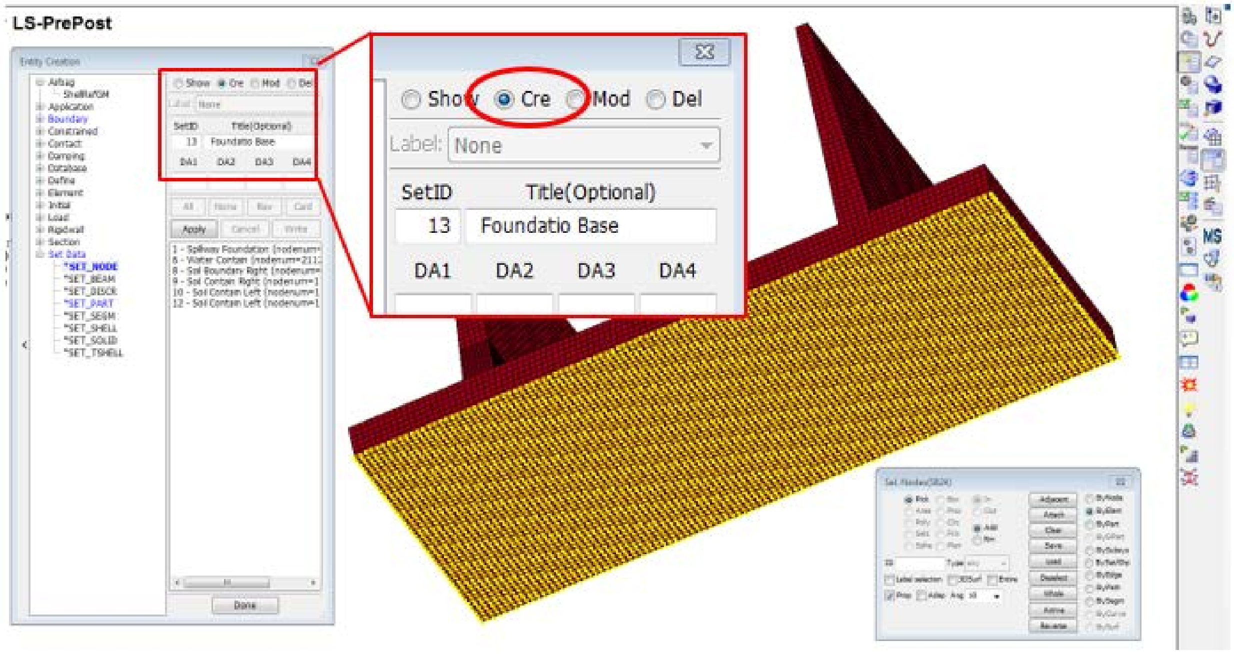


Figure 30 - LS-PrePost "Entity Creation" menu being applied for a "Set_Node".

Sets are used for the contact between the different parts, boundary conditions for the soil elements and boundary for the water elements so the water doesn't flow out of the structure. Please note that creating the Set doesn't assign any of the features (loads, boundaries, or contacts). Creation of the desired feature (e.g. loading) is done in another step. The Set simply creates a group of features that will all be given similar properties. The list of Sets created are shown on the left, like other features created for the model.

Running the Model

Building a model in LS-PrePost will be ready to run in LS-DYNA. To save the model in an LS-DYNA format, select "File," "Save As…," then "Save Keyword As" or (ctrl+shft+s). This will save the model in the text based LS-DYNA (.k extension) format.

To run the program, use the LS-DYNA Program Manager. For instructions on installing the Program Manager and activating an LS-DYNA License, readers are referred to visit http://www.lsdynasupport.com/. In the Program Manager, select the blue button with the down arrow shown in Figure 31, opening the prompt shown. Browse to the .k file as the input file, then "RUN" the analysis. The results can be viewed while the model is solving or following the run by selecting the red button with an arrow making a loop, highlighted in Figure 31.

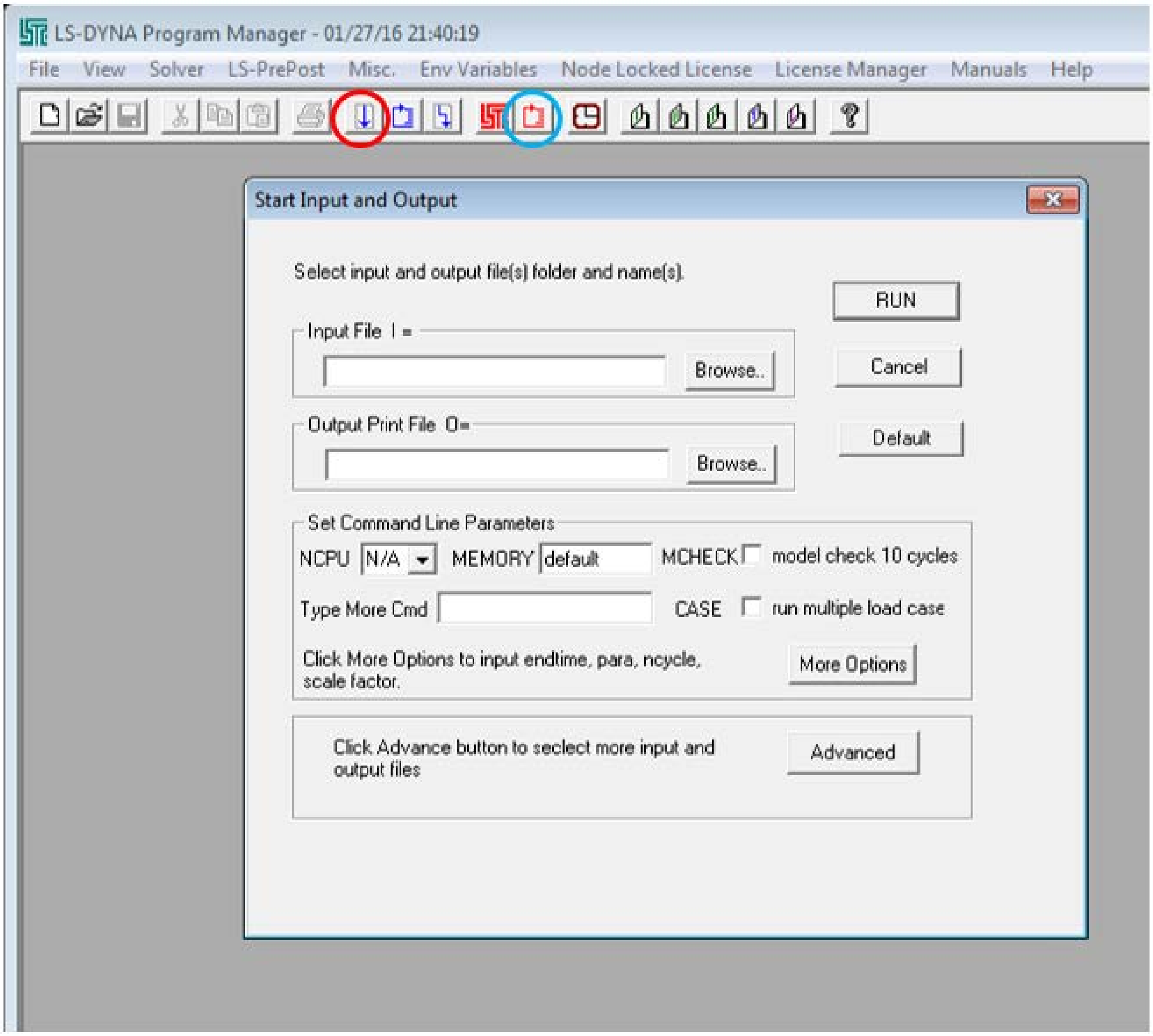


Figure 31 - Running an LS-DYNA .k file through the LS_DYNA Program Manager.

Intro to Post Processing

Post processing of the analysis is performed in LS-PrePost. The “Post Processing” menu can be accessed from them “FEM” drop down menu or on the side bar options. The post-processing options involve plotting the results for specific elements over time (Figure 32), or globally showing stresses at a certain time step (Figure 33), among many other analysis options.

To access the element specific results, such as plotting stress over time, the user will need to access the “History” menu options. To show the results as stress contours on the model at a specific time, the user will need to use the “Fringe Component” menu. Both of these menus are accessed from the “FEM” drop down menu and the “Post Processing” option, or from the menu options on the right-hand side toolbar.

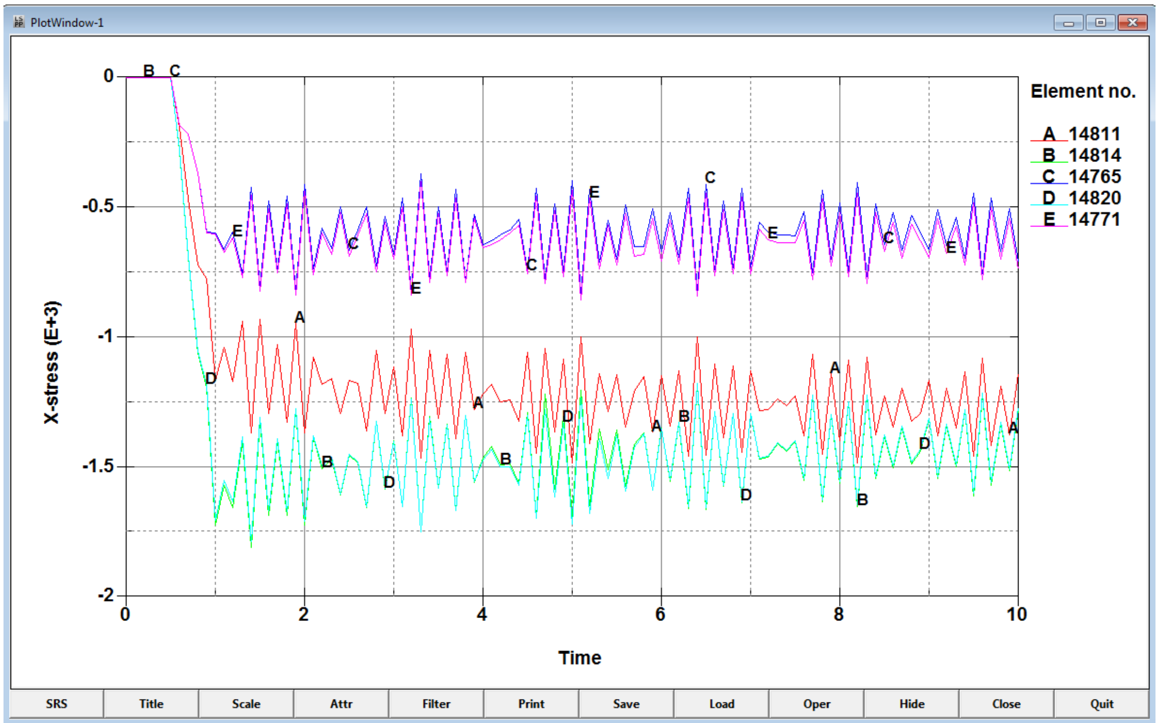


Figure 32 - Stress plot for select elements, over the length of the analysis.

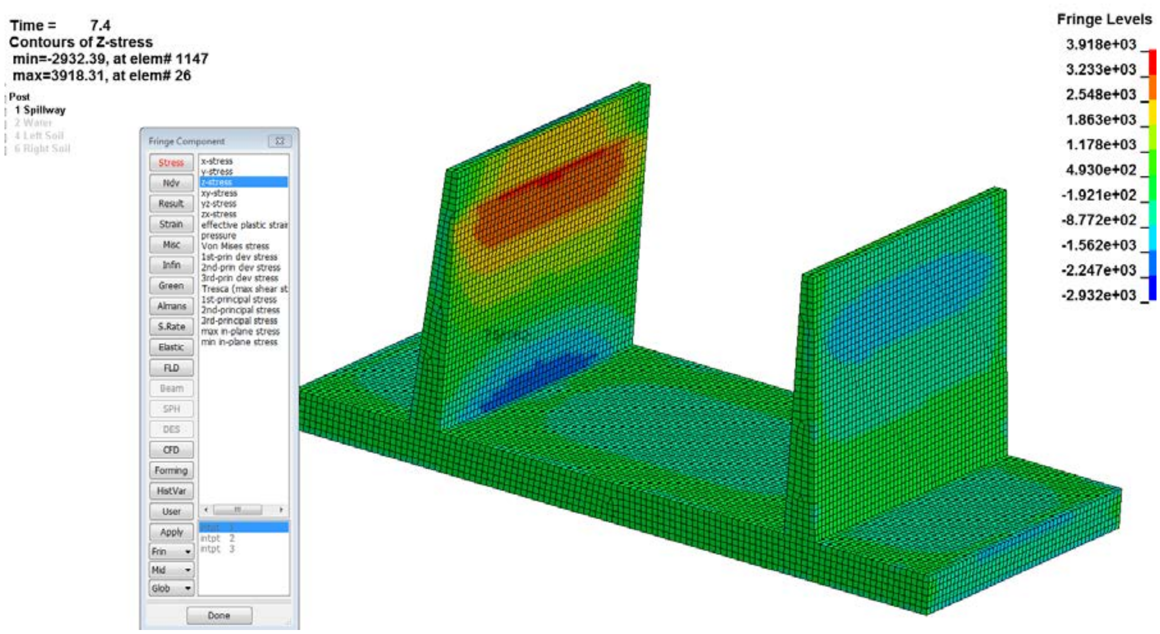


Figure 33 - Stress in the structure at time 7.4 seconds in the analysis.

Advanced Model

Following the Basic Model, a more advanced model of an arch dam was attempted. The dam that was modeled was selected due to the difficult shape to test the limits of modeling within LS-PrePost. The arch dam has a special joint along the foundation and abutments that allows for rotation to dissipate cantilever stresses in the arch. The model consists of the foundation, the arch dam, and the reservoir.

The modeled dam is an unsymmetrical thin concrete arch dam. The dam is "skewed" due to the asymmetrical abutments. The right abutment slopes at approximately 45 degrees and the left abutment slopes at about 30 degrees. The shape of the dam approximates a single-centered double-curvature arch geometry. Building the model started with the foundation, followed by the arch dam, and finally the reservoir.

Foundation Modeling

There are multiple ways to develop a surface foundation, depending on the data that the user has. This research used Civil 3D to generate a surface based on LIDAR survey data, then extracted data from that surface to import into LS-PrePost. From the Civil 3D surface there are two options that were utilized: exporting points on a known grid or exporting contours.

With the points or contours from Civil 3D, a surface model can be generated and meshed into shell elements in LS-PrePost, which will then be converted into solid elements. The steps below assume that the user has already exported the points into a .csv file or the contours into an .iges file. Tutorials for generating points or contours from Civil 3D surfaces are included at the end of this report in Attachment C.

Shell Generation from Points

LS-PrePost can automatically create a mesh from a comma or space delineated file with X,Y,Z coordinates. From the Element and Mesh menu, there is a "Point Cloud to Mesh" option. Figure 34 shows the points loaded into LS-PrePost prior to meshing, and Figure 35 shows the automatically generated mesh. The automatically generated mesh is a triangular mesh.

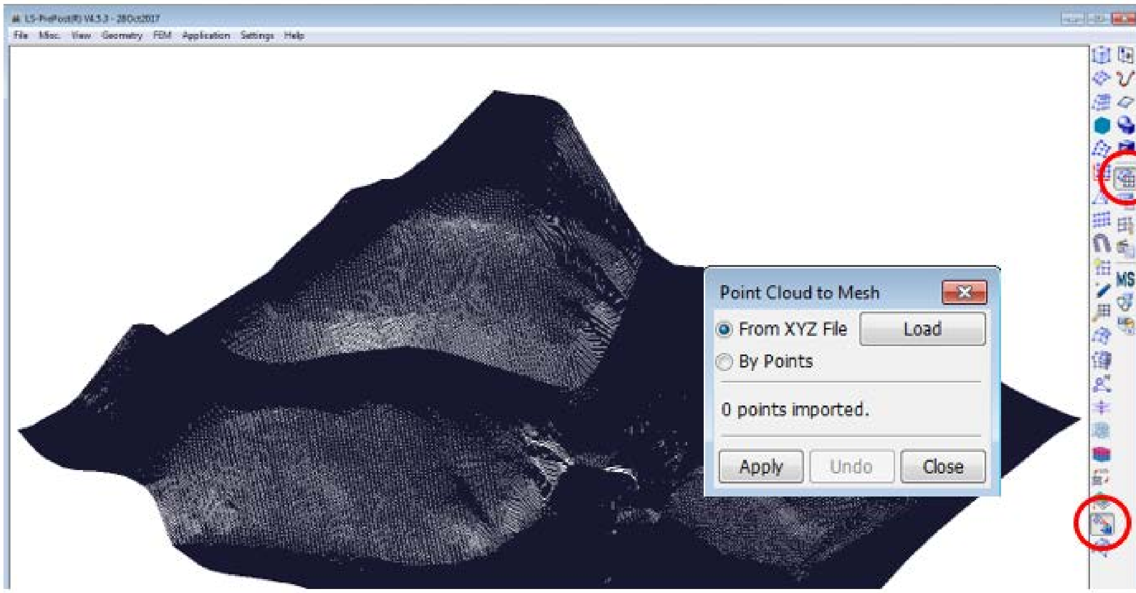


Figure 34 - Imported points from the .csv file.

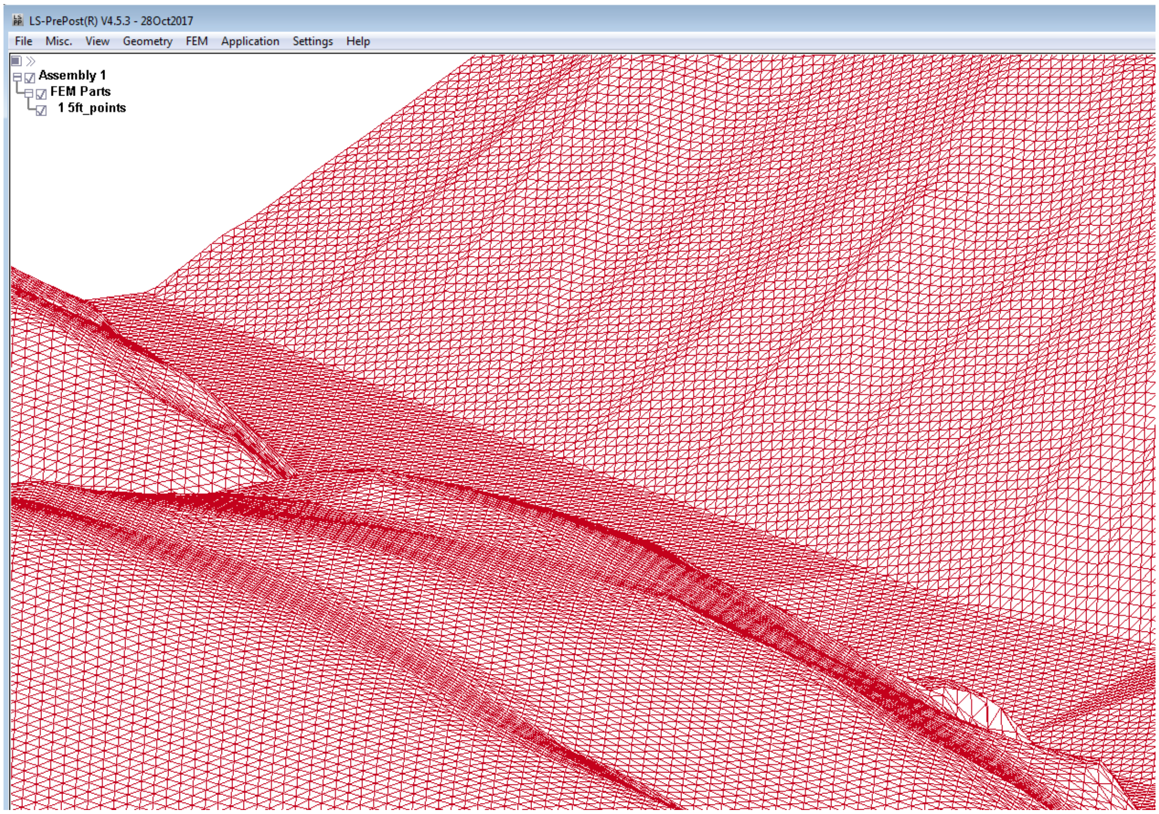


Figure 35 - Close-up of mesh generated from points.

Typically, tetrahedral meshes are not preferred for our solid models, therefore, a conversion to quadrilateral elements was performed prior to converting to a solid model. A triangular shell element will generate tetrahedral solid elements while quadrilateral shell elements will generate hexahedral elements.

To convert the mesh from triangular shells to quadrilateral shells, the Automatic Mesh Generation tool was used. Open the “Auto Mesher” tool in the “Element and Mesh Generation” menu, select “Remesh,” then select all of the elements in the new surface part. Ensure that the mesh type selected is “Quad” to convert to quadrilateral, then select “Mesh,” to re-mesh into quadrilateral elements. Figure 36 shows the Auto Mesher tool and desired options and Figure 37 shows the results of the re-mesh.

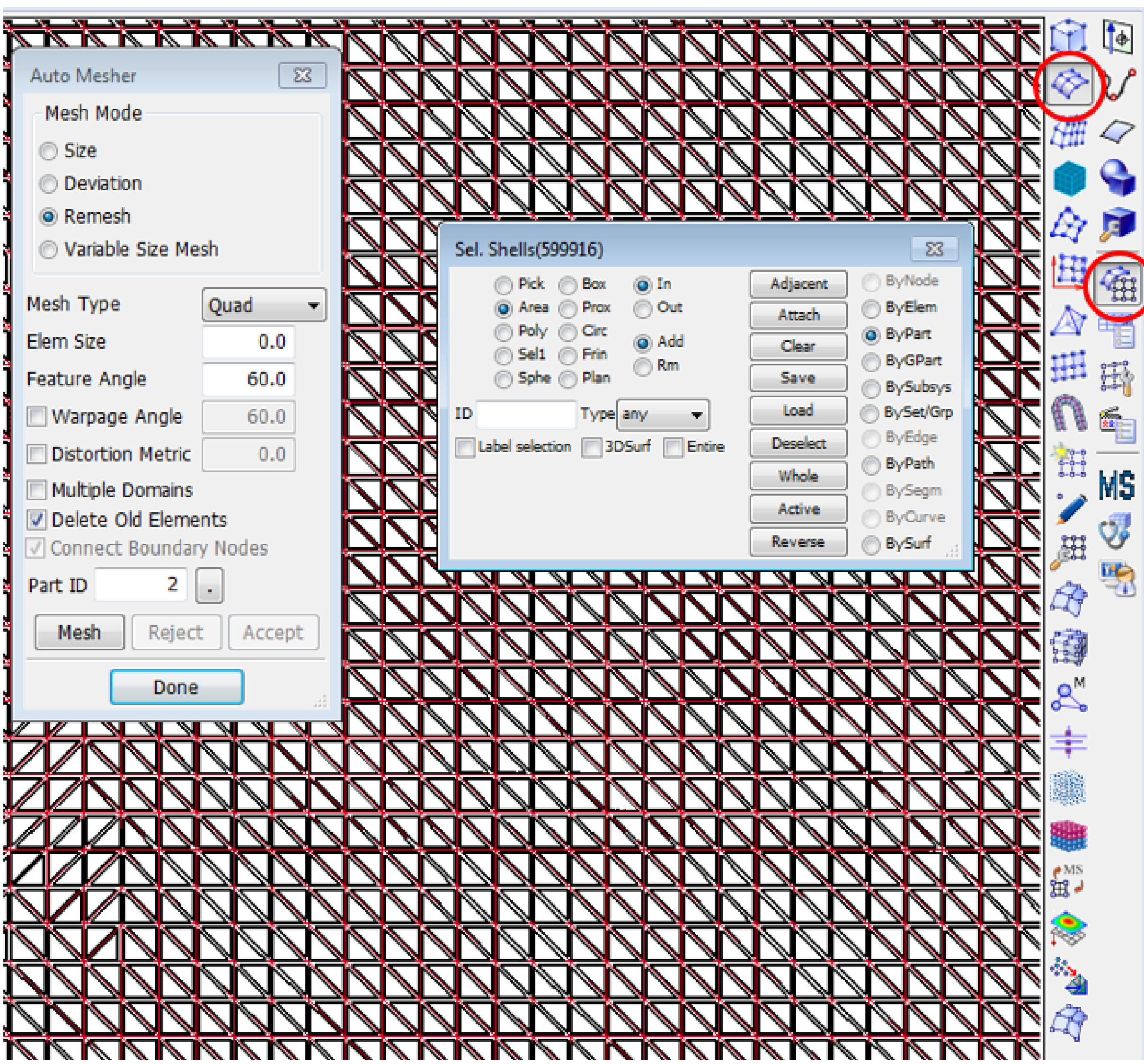


Figure 36 - Automatic mesh generation conversion.

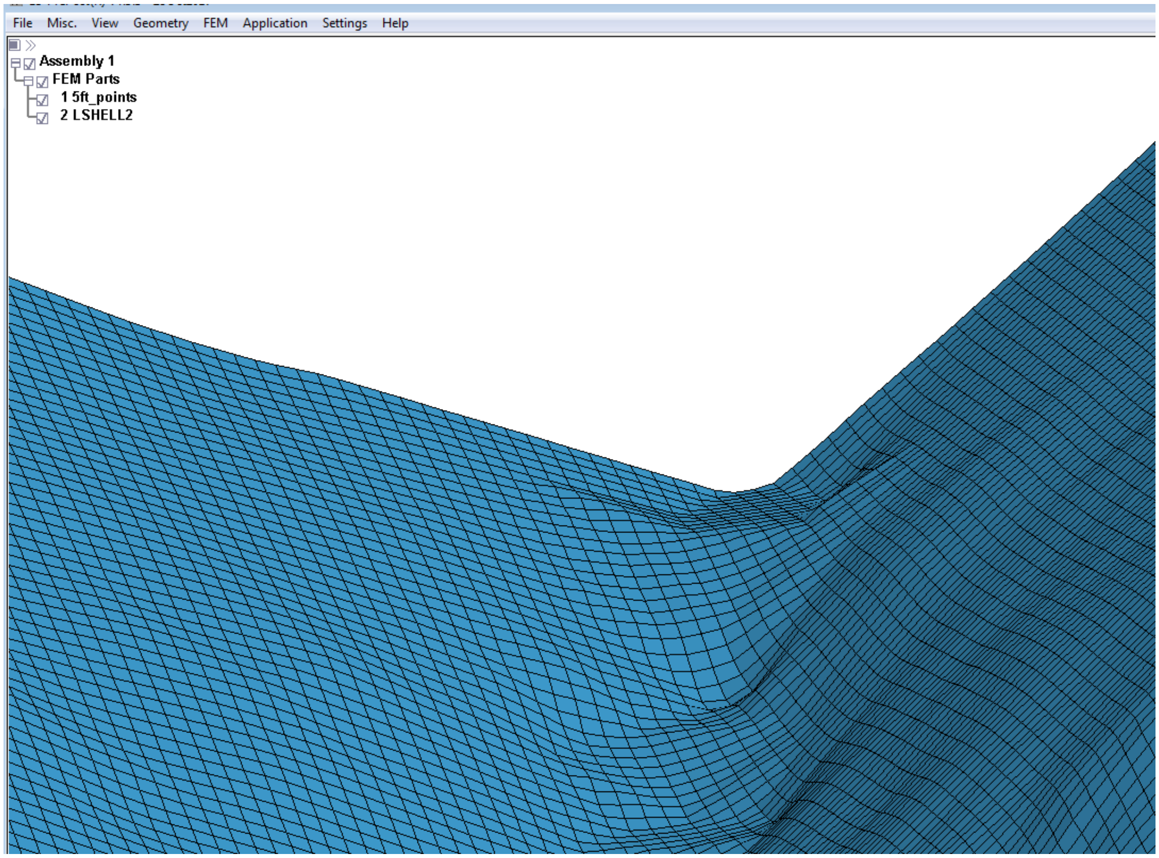


Figure 37 - Result of conversion from triangular mesh to quadrilateral.

To generate the solid elements for the foundation, the shell mesh will be extruded into solids using the Element Generation menu. The quality of the shell mesh will directly affect the quality of the solid mesh, therefore, care should be taken to ensure a quality shell mesh prior to extruding to a solid. It should be noted that the "Block Mesher" may also be used to generate a solid model for the foundation, however that process will be discussed later and the user may decide on which method they prefer.

Shell Generation from Contours

The other option is to generate a surface from contours, which can be meshed into shell elements. First, import the .iges file that contains the contours. Note, the surface generation does not require contours, so 3D curves may also be used. Once the curves have been imported, open the "Fit Surface from Points/Mesh" tool in the "Surface" options. From here you can modify smoothness and fit tolerances, but typically default values will be appropriate. Figure 38 shows the "Fit Surface from Points/Mesh" tool and the preview of the surface generated.

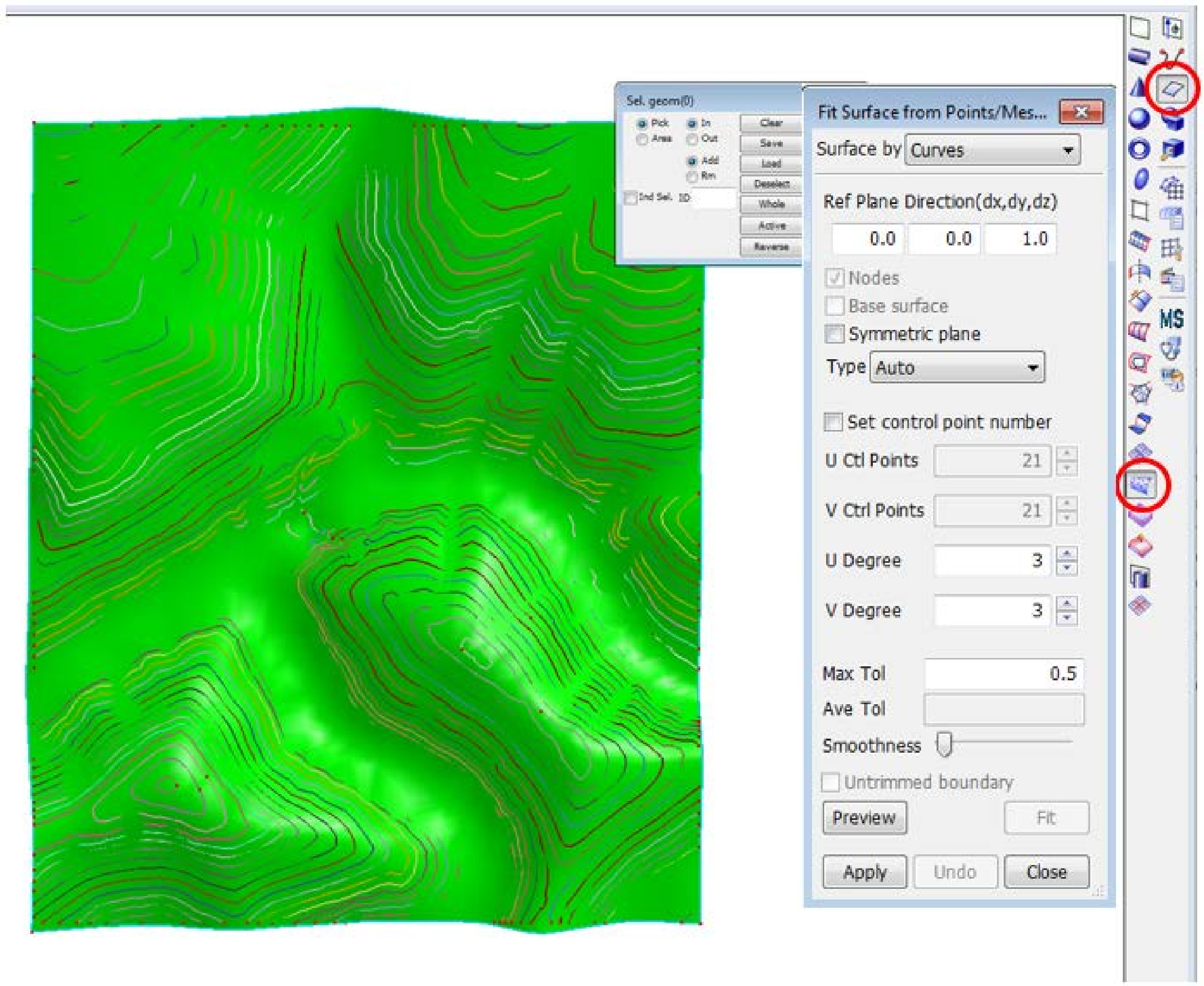


Figure 38 - Surface generation from curves.

After generating the surface, the "Auto Mesher" should be utilized to mesh the surface into shell elements. For the initial mesh, only mixed or triangular elements may be specified. Therefore, the mesh may need to be re-meshed to produce only quadrilateral elements to ensure that extruding the shells into solids will result in hexahedral elements. However, the mesh in this example produced only quadrilateral elements using the "mixed" option. Figure 39 shows a section of the mesh produced from the surface.

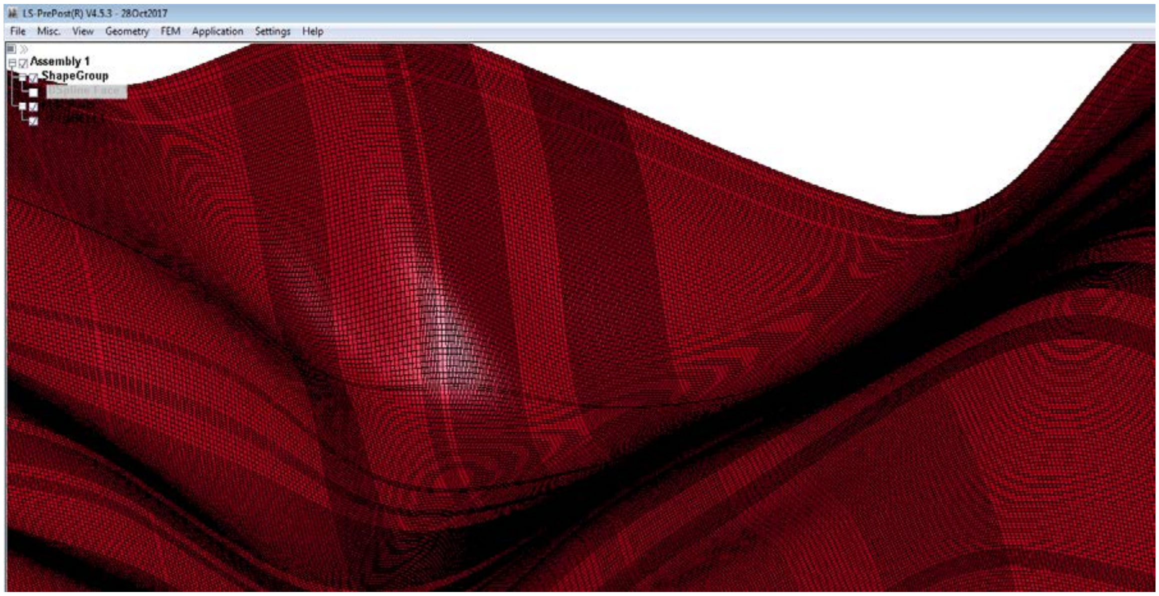


Figure 39 - Mesh produced from curve generated surface.

Solid Foundation Model

Now that the topography has been generated as a shell mesh, it is ready to be converted into a solid part. To convert the foundation shell elements to solid elements the following steps must take place: the surface needs to be copied to the desired depth as a flat surface, then, the shell sweep extrusion method will be used to mesh solid elements between the two shell element parts.

To copy the existing shell part as a flat surface, the “Transform” options in the “Element Editing” menu will be used. In the transform options, select “Project”. Set the projection plane to “Norm Z,” meaning the part will project normal to the Z plane. Next, select the “Pick Location” option then any node for the “NodeID” and the corresponding location will autofill the “XYZ” field. Change the “Z” coordinate to the desired model base. Then, specify the vector for the transformation by selecting the “Z” button, or typing “0,0,1” in “X,” “Y,” and “Z.” Make sure “Copy Elem” is checked so that a copy is made and that a new part number (2 in this example) is specified, otherwise these elements will be added to the same part and the solid element generation will not work. Finally, select the entire part to copy the part, and then select “Project.” Accept the new part if it is in the desired location. Figure 40 shows the projected shell part below the existing part as well as the “Transform” menu.

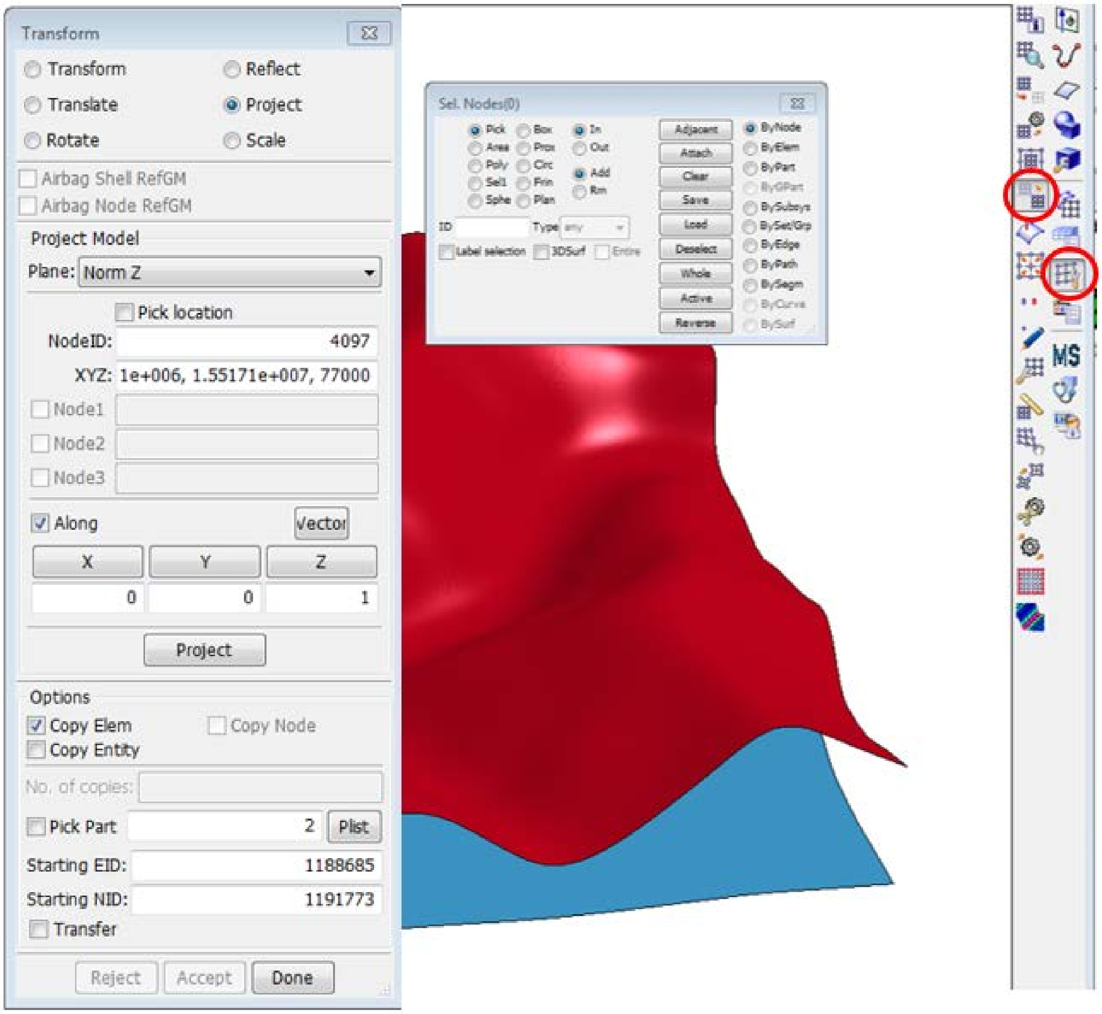


Figure 40 - Transform menu to duplicate shell part as a flat part.

Now that there are two parts with an equal shell count, the “Element Generation” tool in the “Element and Mesh” menu can be used to generate solid elements between the shells. This is the same tool that was used in the basic model with the Shell Sweep option. Opening the “Element Generation” tool, select “Solid” option at the top, then specify “Solid By” as “Two_Shell_Sets.” The next step is selecting the Shell Sets, and this is done by specifying two corresponding nodes on each Set, however, the order of this is important for correctly performing the extrusion. The following steps should be performed in order, with the final result shown in Figure 42:

1. Select the box “Set1” even though it is already checked
2. In the “Sel. Shells” box, change the selection type to “ByPart”
3. Select either part
4. In the “Element Generation” tool, select “Set1” “N1”, then select an easily identifiable node (e.g. a corner node) from the part specified as “Set1”.
5. Repeat step 4 for “Set1” “N2”
6. Repeat steps 1 through 5 for the part that wasn’t selected. Ensure that N1 and N2 from each part correspond (see Figure 41).

7. Specify the number of segments – this will be the number of solid elements, minus one, that make up the depth of the foundation.
8. If desired, specify the bias. This is weighted toward "Set1." A higher number will space the segments further from "Set1," a smaller number will space the segments toward "Set1," and a value of 1 will have equally spaced segments.

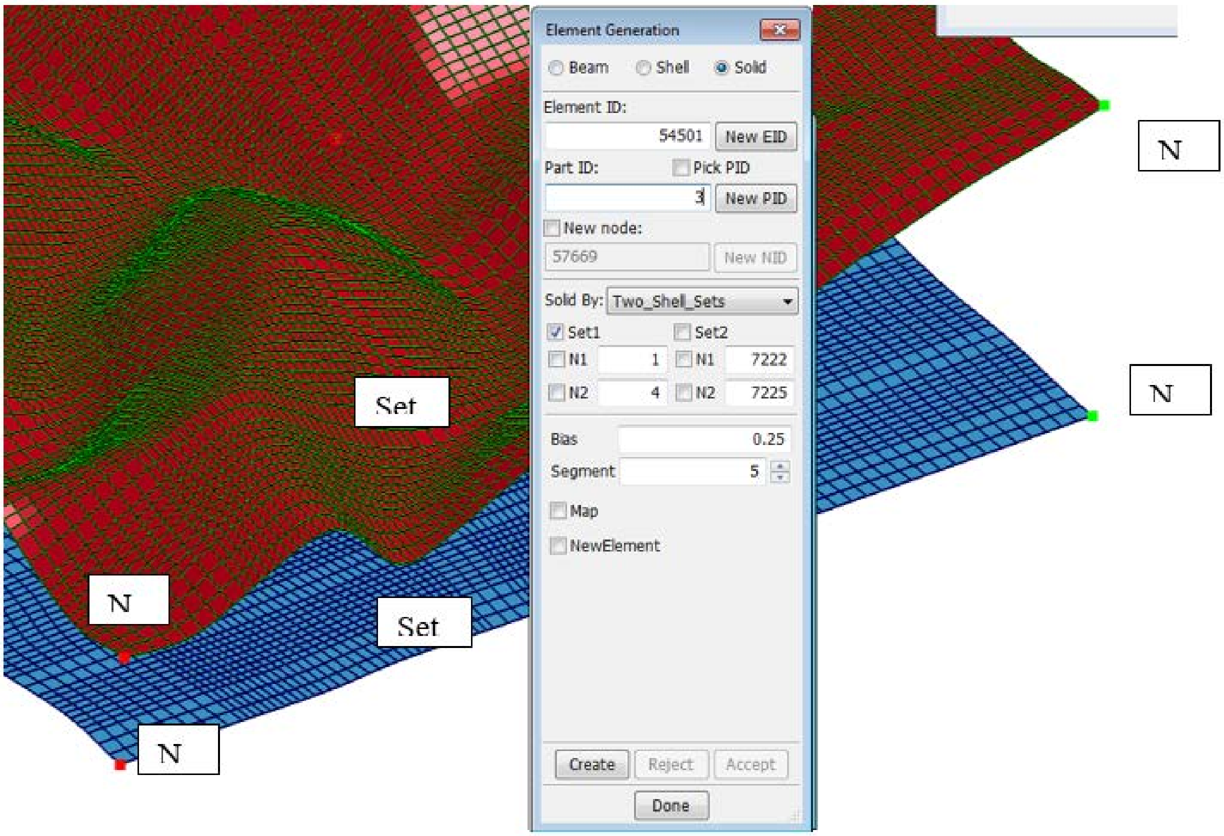


Figure 41 - Element Generation Two_Set_Sweep.

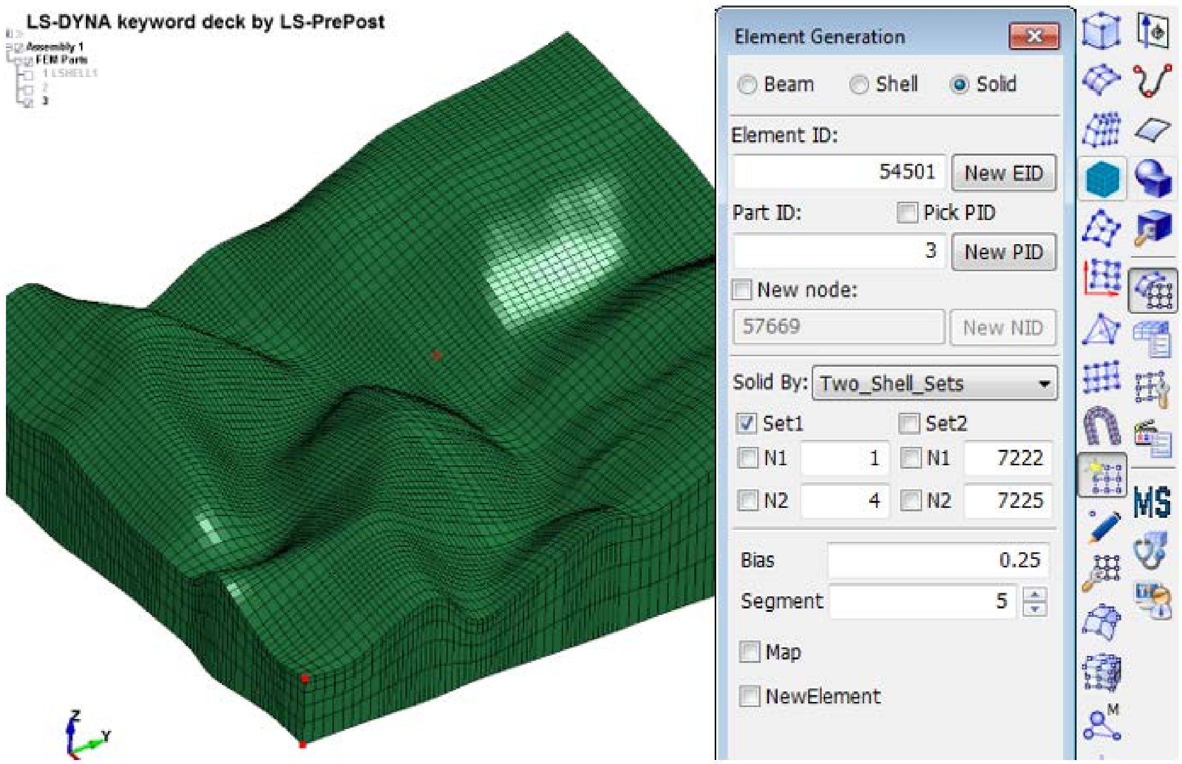


Figure 42 - Resultant solid foundation model from Element Generation Two_Set_Sweep.

Modifying Mesh

In some instances, the foundation mesh (or any mesh) may need to be refined in a specific area of the model. In this case, the mesh under the structure was modified to be a finer mesh. The easiest way to do this is to modify the shell mesh prior to creating the solid mesh.

Modifying the mesh and retaining quadrilateral elements is a two step process. The first step is to use the Element Editing tools to split the shell elements. Figure 43 shows the selected shell elements to split as well as the Element Editing tool with the Split options shown. Under the "Split Operation: Shell" section, checking the first box splits elements into quadrilateral elements while the second box splits elements into triangular elements. Additionally, selecting the "Retain Bdy" box ensures that the boundary nodes are retained, however this may cause triangular elements along the boundary (Figure 44).

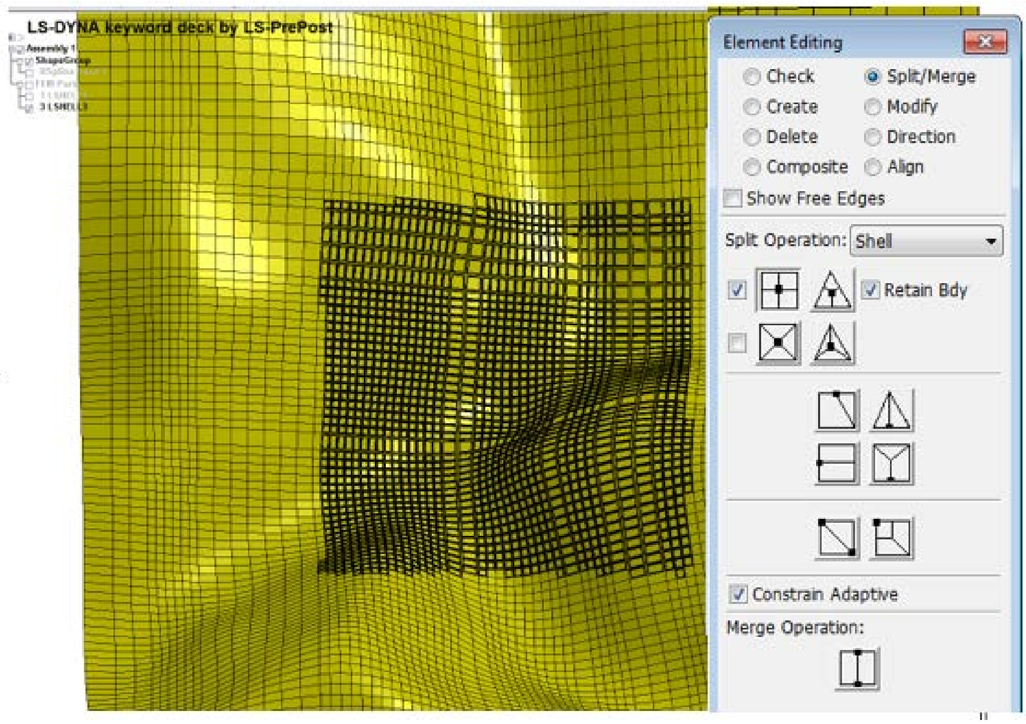


Figure 43 - Element Editing tool showing shell split options.

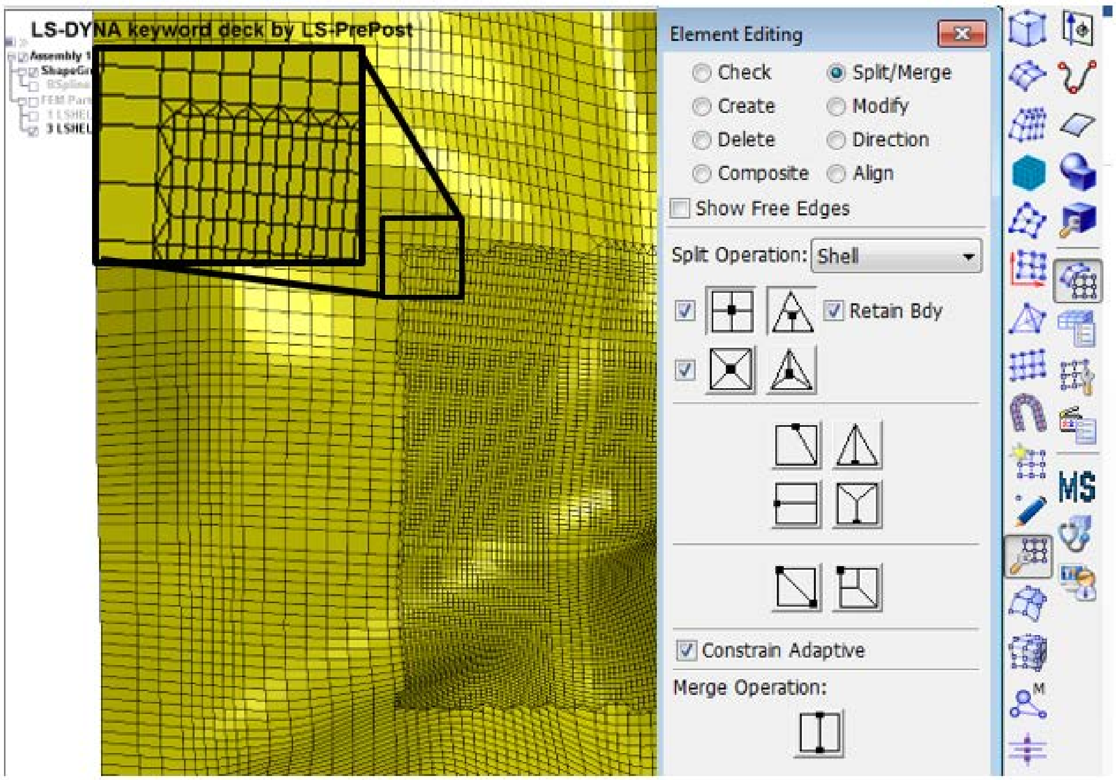


Figure 44 - Results of shell splitting operation.

Following the splitting operation, it may be necessary to use the Auto Mesher to remesh the elements into all quadrilateral elements. To do that, open the Auto Mesher and select all of the modified elements as well as the boundary elements. Then, select the “Quad” Automesh option remesh. Figure 45 shows the remesh in pink.

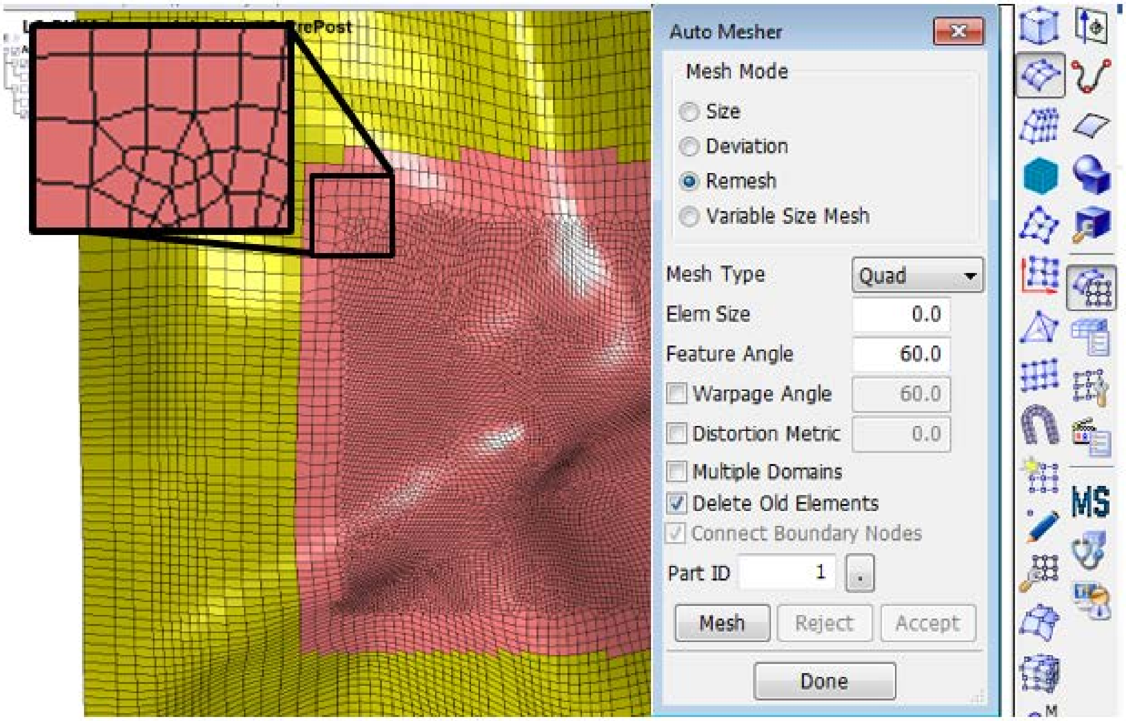


Figure 45 - Automatic meshing of split shell operation.

When the mesh is completed, return to the "Solid Foundation Model" steps to create the foundation model with solid elements.

Structure Modeling

In the basic model, 3D objects were imported from AutoCAD and meshed directly in LS-PrePost using various meshing methods. However, complex 3D shapes do not mesh as hexahedral elements as easily as basic shapes. If a tetrahedral mesh is acceptable, the tetrahedral meshing tool is capable of meshing complex shapes accurately (see Figure 46) and very quickly. Caution should be applied when attempting the Solid Mesher. At this time, it is believed that the Solid Mesher does not work for complex shapes.

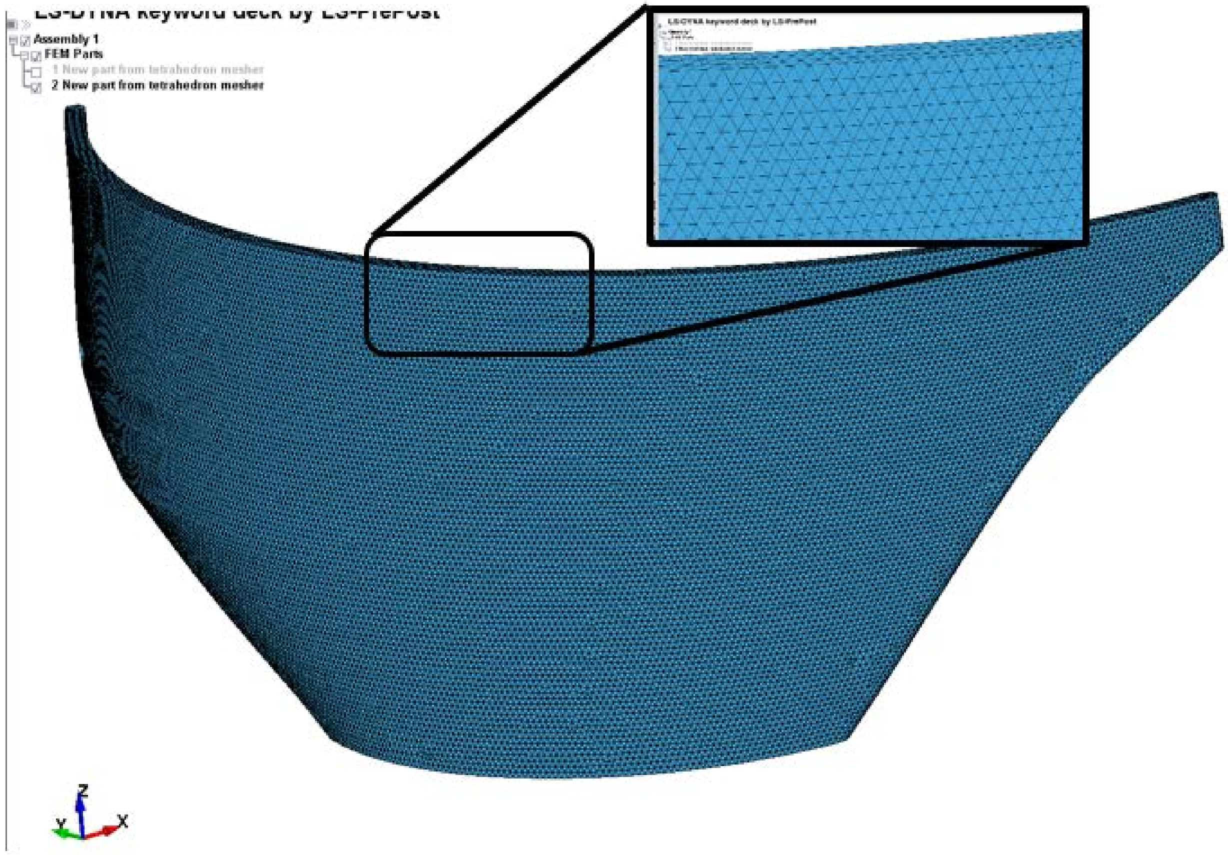


Figure 46 - Tetrahedral mesh of an arch dam.

If a tetrahedral mesh is unacceptable, combining the N-Line Mesher with a cross section and performing a “Shell Sweep”, similar to the Basic Model, will work well for complex shapes of similar cross sections. But, if the geometry changes in multiple directions, breaking the shape into cross-sections is easiest, utilizing the “Two_Shell_Set” method, similar to the foundation modeling.

In this case the N-Line Mesher with the “Two_Shell_Set” method was used. To make an arch dam solid, the dam was split into cross sections at heights of 10-feet, based on information given in the drawings. Curves were imported into LS-PrePost representing each cross section and the N-Line Mesher was used to make similar meshes for each cross section, see Figure 47. Once all of the cross sections were meshed, each section was extruded, using the “Two_Shell_Set” method, to the next section. However, in the “Two_Shell_Set” input, keep using the same part number for the extrusions. This will combine the extrusions into one part, see Figure 48. Unless the block editor is being used, you cannot project solid elements to surfaces, therefore, be sure to include details such as the curved sides of the arch, if required.

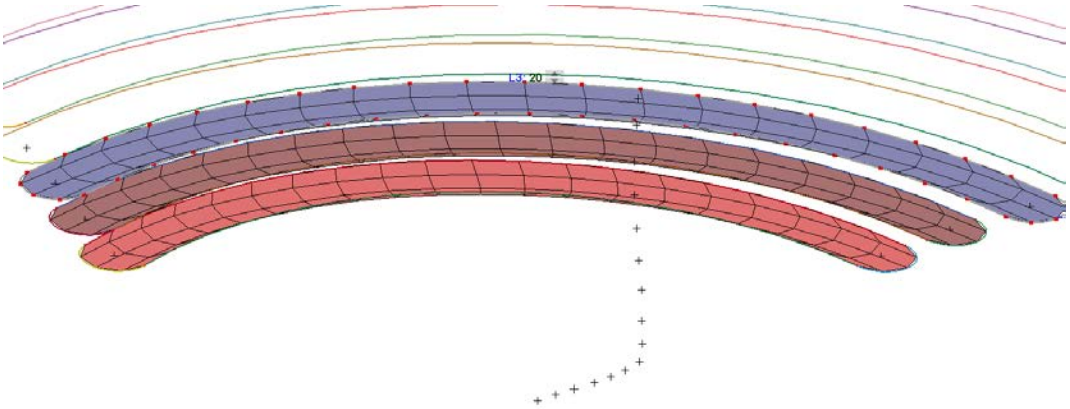


Figure 47 - Cross section meshing using N-Line Mesher.

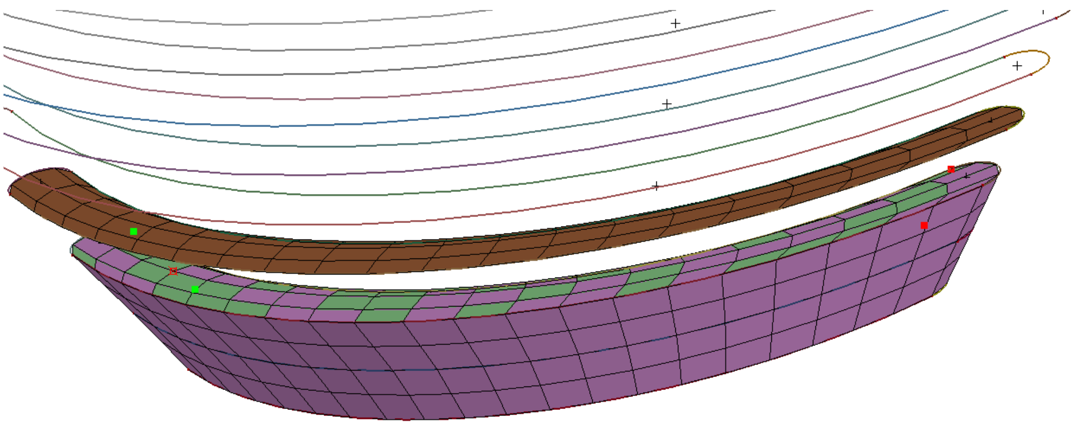


Figure 48 - Converting shell parts to one solid part using the Element Generation Two_Shell_Sets feature.

Following completion of the arch, use the “Element Editing” tool to align all of the solid elements in the same direction. Using thing Two_Shell_Set method appeared to randomly orient the solids, which will affect how local results are read, or how solid modification processes occur. To align the solids, use the “Element Editing” tool from the “Element and Mesh” menu. Select the “Align” option, then select the type of elements that should be modified; solid in this case. Select any element from the part then select align; this will align all of the elements in the part. Additionally, there are options to show normal arrows or show the element direction to verify the element directions, as shown in Figure 49.

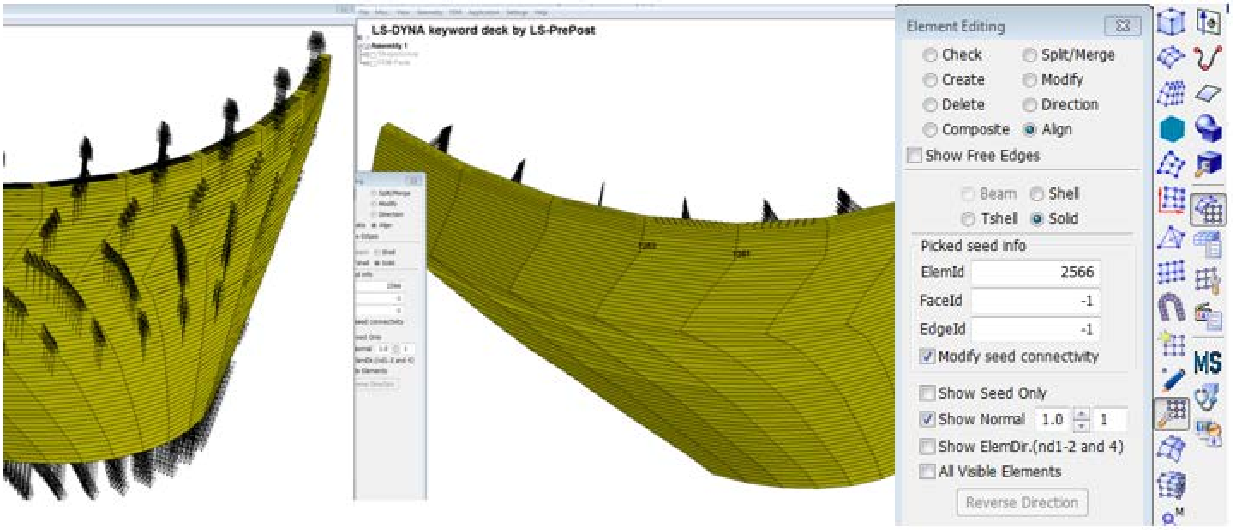


Figure 49 - Element normal direction before (left) and after (right) aligning to element 2566.

Following alignment of the solids, the user may split the solids into a finer mesh (if needed) using the "Element Editing" menu and the "Split/Merge" option. Figure 50 shows the solid split operations and before and after figures of the arch part. Caution should be used when splitting the solids; if the solids are oriented in different directions the solids will not all be split on the same plane. Additionally, following a split, the new solids may not follow the same alignment and, as a result, a more iterative process of aligning followed by splitting is recommended.

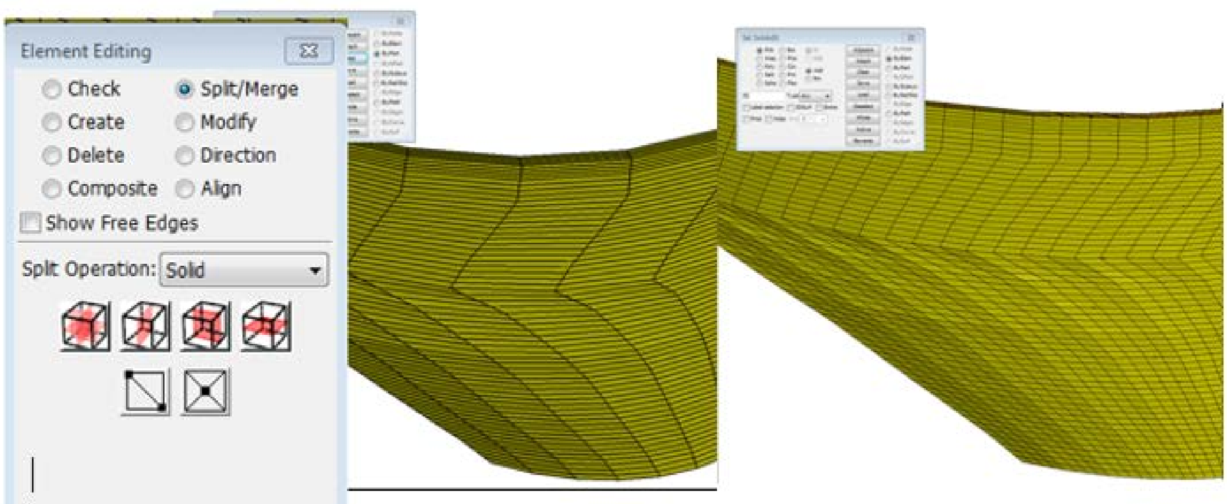


Figure 50 - Splitting solid elements before (left) and after multiple iterations (right).

Modeling between Parts – Reservoir Modeling

Parts in models may interact with one another on surfaces that are not easily defined by simple shapes, such as a reservoir on the foundation surface. If both shapes had flat boundaries, placing the parts against each other would be easily done, however, since a foundation surface is typically non-uniform in any direction, a part in contact with the foundation would need to be modeled using the foundation as a surface to mold to. This can be done in LS-PrePost if a block is created using the “Block Mesher” tool in the “Element and Mesh” menu.

The Block Mesher tool in LS-DYNA generates a new single or multi-block part, then allows the user to transform that part with a variety of functions within the Block Mesher. The Block Mesher is the only known tool that will allow the user to project a solid part to another surface, which is critical for reservoir modeling. A reservoir is contained on all sides except the top, forming to the objects containing it: the foundation and the dam, as well as any dikes.

First, surfaces need to be developed for the foundation and retaining structures. Figure 51 shows the boundary elements for the advanced model. Start by opening the Block Mesher tool in the “Element and Mesh” menu. Within the Block Mesher the user must first define a new block type: Multiple Blocks, Single Block, or Butterfly Blocks. For this example, we will use a Multiple Block type. Approximate the location of the block; for the multi-block part the data must be manually entered, however for the single block part the user can select nodes to define the block. After defining the block, select “Create,” and the block will appear on the screen with a separate “BlockM Computation Window,” shown in Figure 52.

The “BlockM Computation Window” is an interactive window that allows the user to select faces or blocks to then modify individually. Figure 53 shows block 1 1 1 2 2 2 selected in the “BlockM Computation Window.”

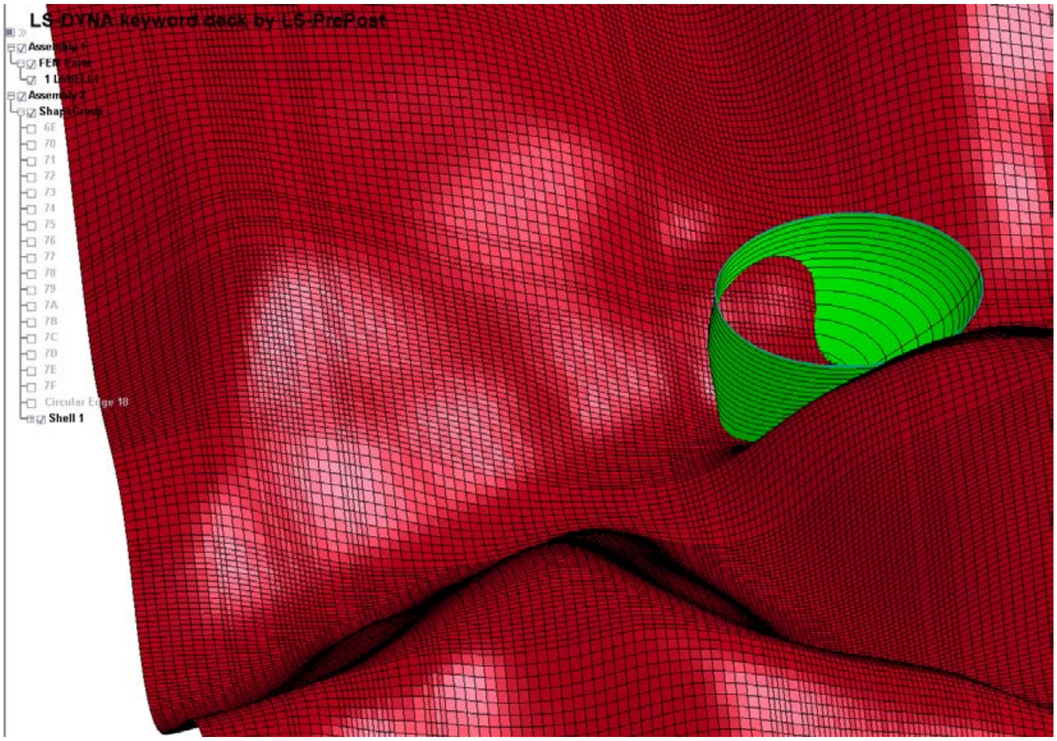


Figure 51 - Boundary surfaces for reservoir part.

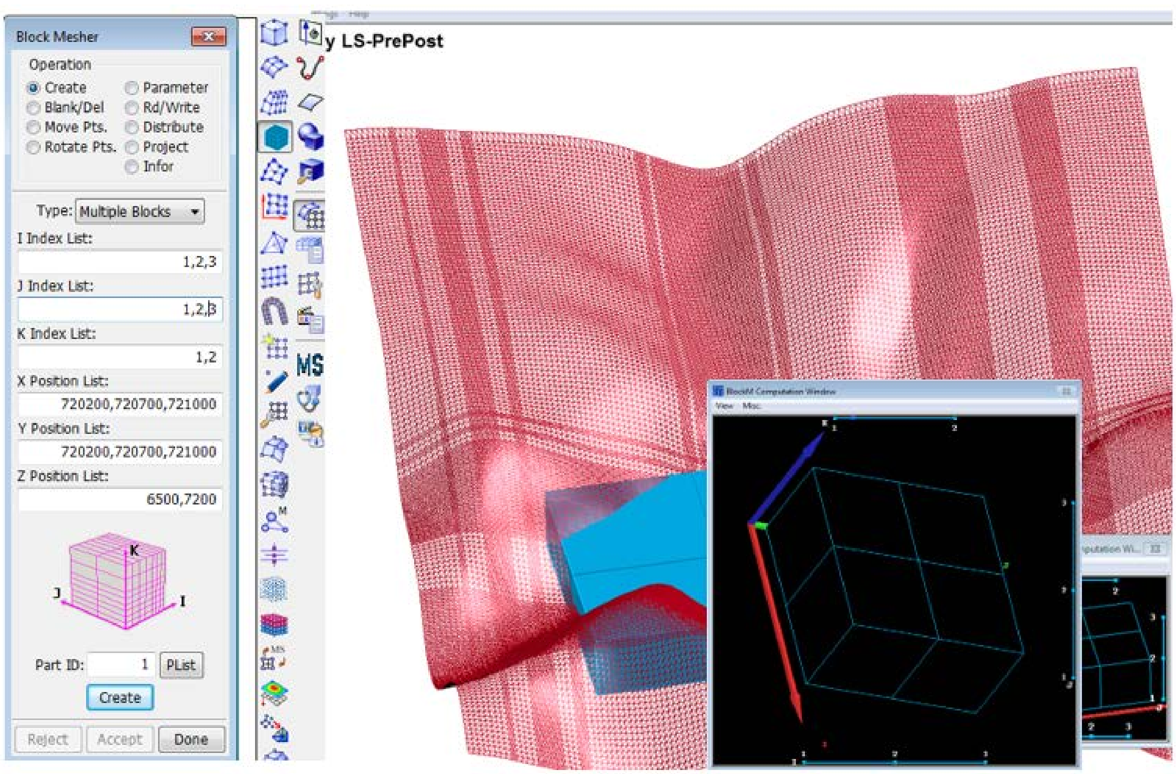


Figure 52 - Block Mesher tool with BlockM Computational Window and the initial block generation.

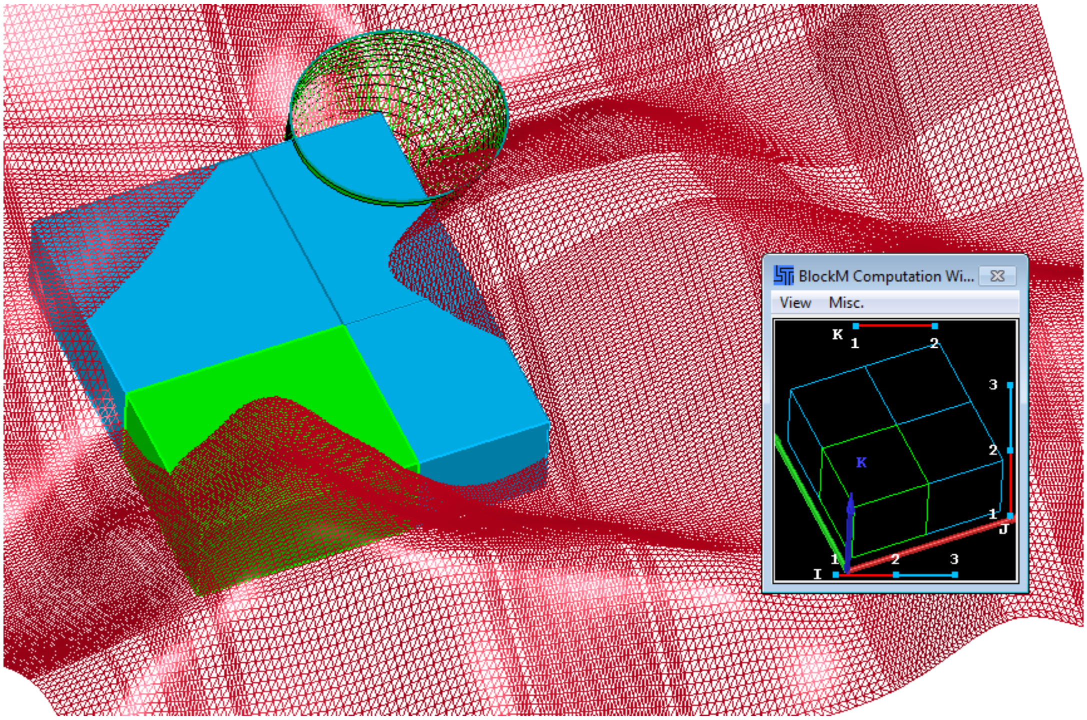


Figure 53 - Example of using the BlockM Computation Window to select a block.

Use the different operations in the “Block Mesher” tool to modify the block as desired. One of the more useful options is the “Project” operation that will allow the user to project part of the block(s) to a surface, shell, or other feature, however, the projection is static, and once something is moved the projection needs to be completed again. Therefore, the last operation completed should be projections. Figure 54 shows the reservoir block (yellow) projected to the foundation and the cylinder representing the arch.

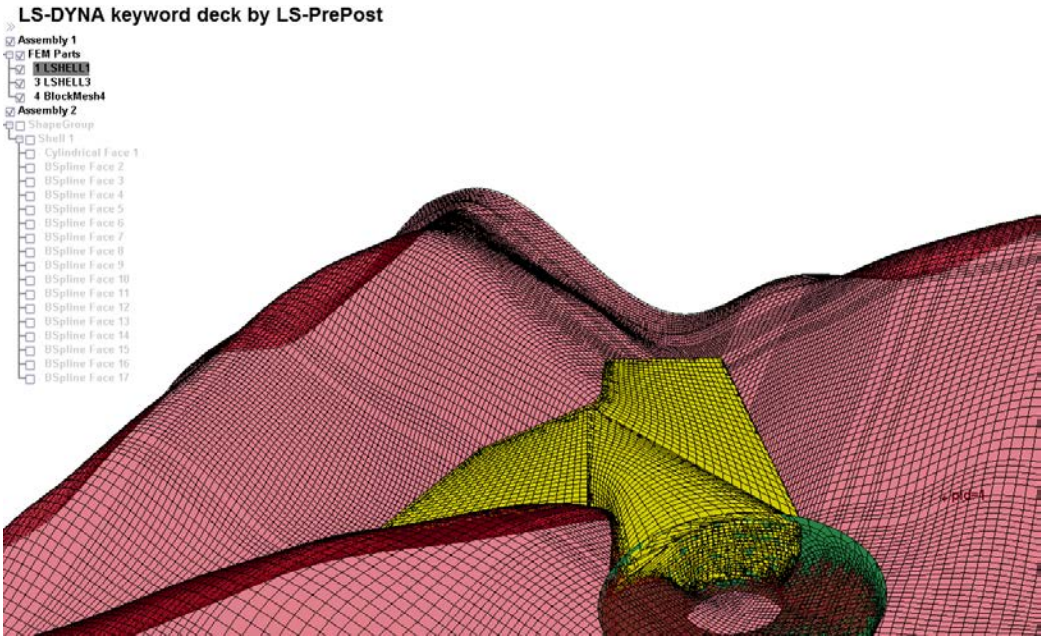


Figure 54 - Reservoir block created using "Block Mesher."

The “Block Mesher” is a capable tool; however, it is initially difficult to use and requires expertise to ensure model accuracy. Multiple iterations typically need to be attempted, and there is no “undo” option in LS-Prepost, so it is important be sure to save often. The user is encouraged to attempt the different options and projection directions to piece their parts as desired.

Checking the Model

It is known that an accurate FEA the quality of the mesh is critical. One of the benefits of using LS-PrePost to develop the model is that there is a built-in model checking tool, shown in Figure 55. The Model Checking tool is accessed in the “Application” drop down menu or from the toolbar on the right of the GUI as a stethoscope symbol.

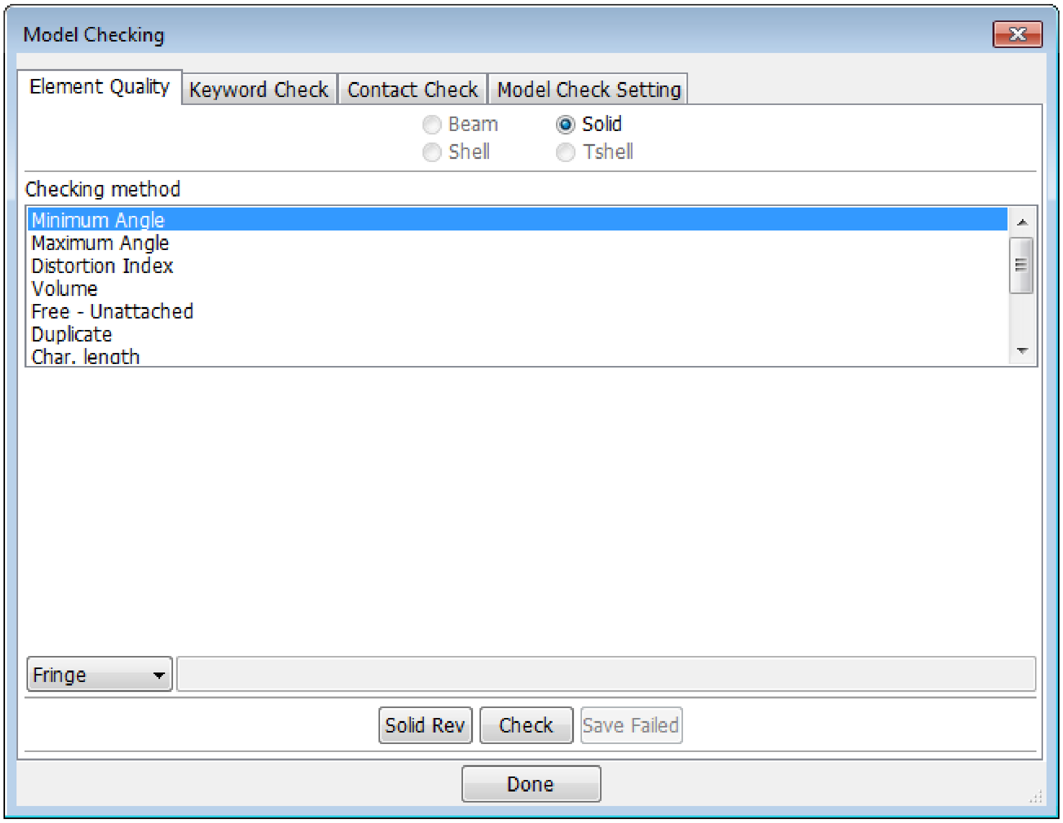


Figure 55 - Model Checking tool in LS-PrePost.

After a check is performed, the model will update to highlight elements that do not meet specified criteria. For example, Figure 56 shows a check on the aspect ratio of the solid elements making up the reservoir, with blue representing elements with an aspect ratio of approximately 1 and red representing elements with an aspect ratio around 9.

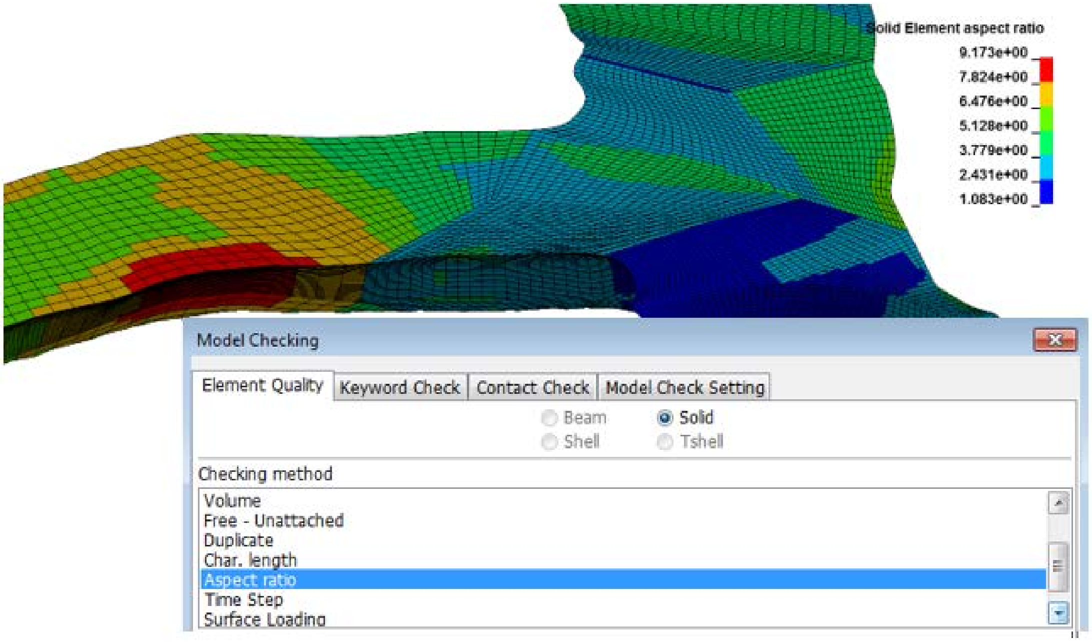


Figure 56 - LS-PrePost check of element aspect ratio.

Based on the initial check, it was observed that the elements on the left-hand side of the reservoir had poor aspect ratios. Thus, those elements were isolated using the “Blank Mesher”, which allows the user to blank out elements to view hidden elements (Figure 57). Once the elements were isolated, the remaining elements were divided along their long axis using the “Element Editing” Split/Merge Solid editing operations (Figure 58). The next iteration of the aspect ratio check is included as Figure 59.

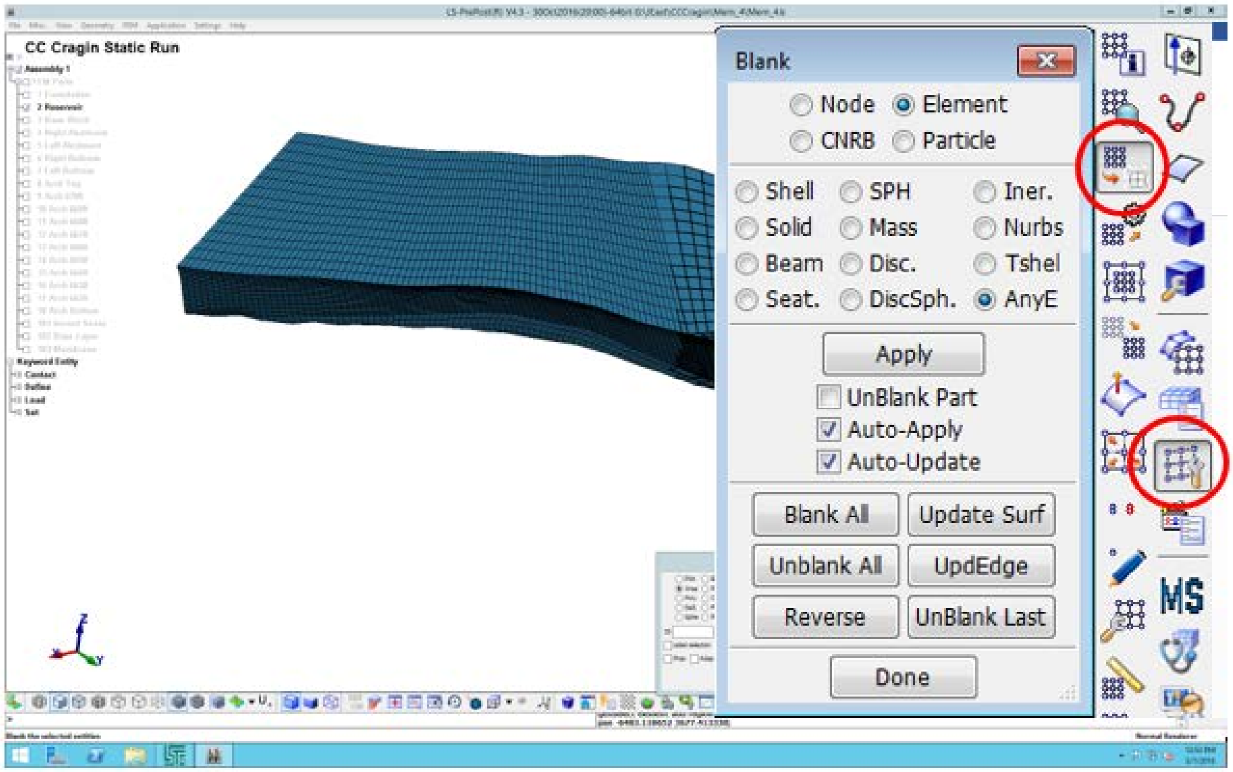


Figure 57 - Blank tool prior to applying the changes. Figure 58 shows the blank applied.

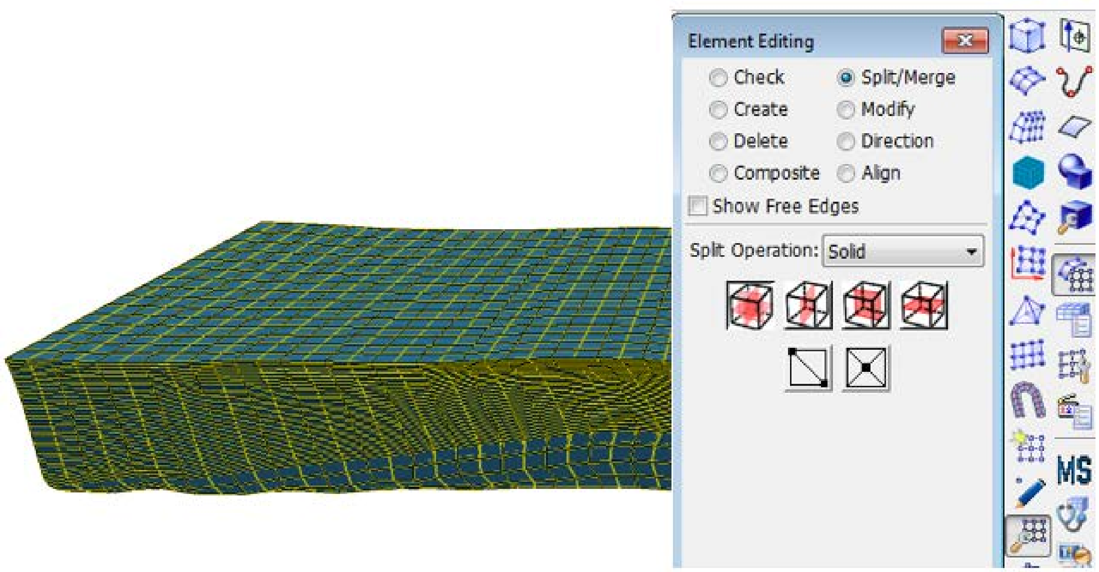


Figure 58 - Isolated solid elements being divided using Element Editing tool.

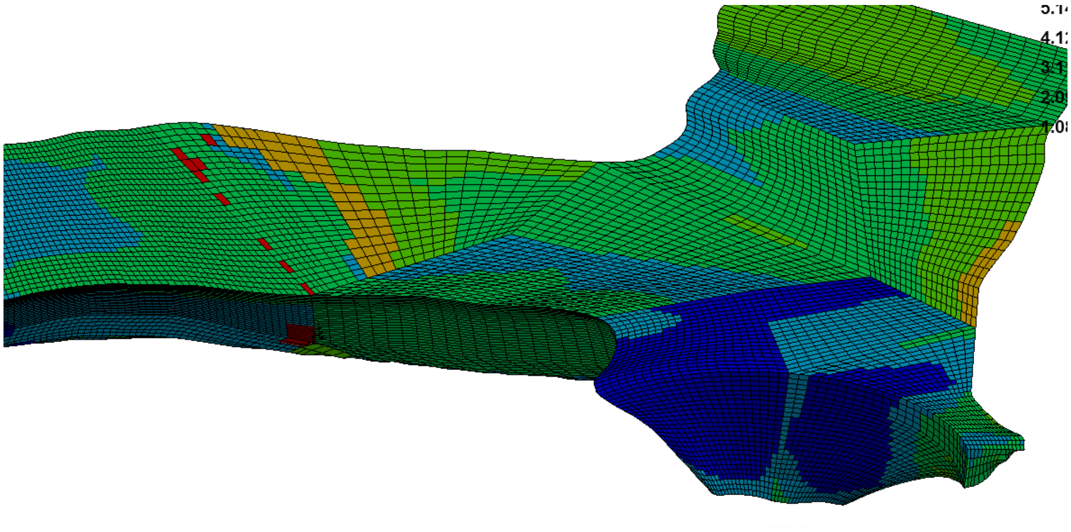


Figure 59 - Aspect ratio check after dividing the elements.

Correcting the model is an iterative process, using a variety of the checks, especially if hexahedral elements are required. However, if a tetrahedral mesh is acceptable, one or two iterations remeshing with the Tet Mesher will typically work, especially since tetrahedral elements can adapt to different geometrical shapes much easier. Following the first check the reservoir was also remeshed using the Tet Mesher to correct the aspect ratio, as shown in Figure 60. Please note, a mesh cannot be converted back to a hexahedral mesh after being converted to a tetrahedral mesh.

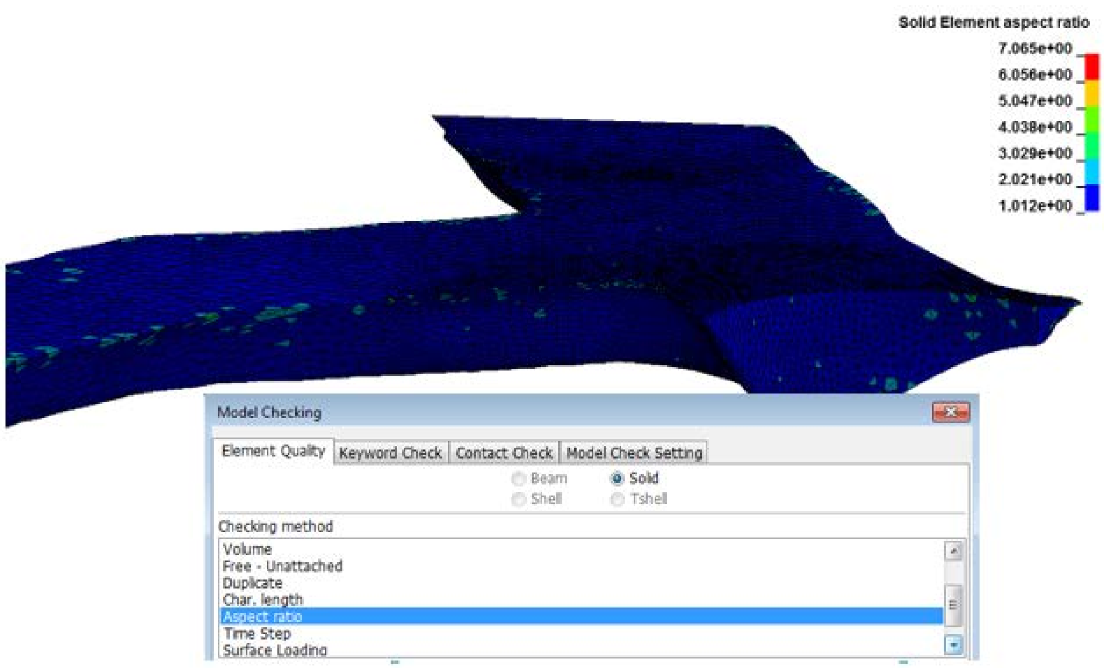


Figure 60 - Aspect ratio check following remeshing with the Tet Mesher.

Concluding the Advanced Model

In the advanced model, the foundation, dam and reservoir were created. Contacts, loads, boundary conditions, etc., assignments are the exact same as highlighted in the basic model. The intent of the advanced model was to determine the capabilities of modeling the structure in LS-PrePost.

Conclusion

This report documents a procedure of using LS-PrePost to develop a FE model and serves as a learning tool and tutorial for LS-PrePost. There are many ways to develop a model in LS-PrePost and not all of them were covered in this tutorial. However, this tutorial should give the reader a solid basis to start using LS-PrePost for modeling, which will allow the reader to explore alternative methods for modeling in LS-PrePost and determine the most appropriate way to develop their model.

Recommendations for Future Research

This research relied heavily on LS-PrePost as a modeling tool, using AutoCAD and other Autodesk products to only generate the shapes to import. However, the research did not consider the possibility of meshing an object within the CAD software used to develop the models. If an entire model could be generated in a CAD program, then LS-PrePost could be used only for developing the LS-DYNA input codes (contacts, boundaries, etc.). However, this would also introduce another program and another source of error in modeling when converting from one program to the next. The benefit of modeling in CAD is that many engineers would be able to develop models for FEA, instead of only those that are familiar with FEA specific programs.

Further, there are many different preprocessing programs available to develop finite element models. It is likely that each program has similarities, as well as distinct advantages and disadvantages over other programs. Future research could look into different preprocessing programs for development of FE models and identify the advantages and disadvantages of each program to assist in analysts choosing the most appropriate program for each specific job.

Currently, research is being conducted at Reclamation on using drones to collect LIDAR data to develop 3D models. Another area of future research should consider taking the LIDAR data 3D models one step further and possibly using LS-PrePost to transform those 3D models into FE models. The difficulty with converting LIDAR 3D models to FE models is the significant amount of data provided by the LIDAR 3D scans and reducing it to something usable within LS-PrePost without significantly diminishing the quality.

References

[1] K.-J. Bathe, "Finite Element Procedures in Engineering Analysis," Prentice-Hall, Inc., Englewood Cliffs, 1982

[2] MIT Open Course Ware topic on Finite Element Procedures for Solids and Structures," MIT, Cambridge.

[3] Salamon, J. Presentation to group, “Mathematical formulation of an engineering problems”, Electrical file Q:\Civil Engineering\D8110\Z-OTHER_8110 Group learning\Intro to nonlinear FEA – Internal 86-68110 TSC Lecture.

[4] “Revit”. Vers. 2016. Sausalito, CA: Autodesk, 2016. Computer software.

[5] “Inventor”. Vers. 2016. Sausalito, CA: Autodesk, 2016. Computer software.

[6] Qasim, S., Abid, H. “LS-DYNA for Beginners,” Lap Lambert Academic Publishing GmbH KG, 2012.

[7] Wang, Erke et. al. “Back to Elements – Tetrahedra vs. Hexahedra,” CAD-FEM GmbH, Munich, Germany. Accessed November 27, 2017.

[8] “LS-DYNA Keyword User’s Manual,” Version 970, Livermore Software Technology Corporation, April 2003.

ATTACHMENT A

AutoCAD Shell Model Development (LOFT)

Setup AutoCAD for 3D Modeling

The first step to 3D modeling in AutoCAD is the setup access to the 3D modeling tool. To start, right click on the "Ribbon" in AutoCAD. Then, hover the mouse over the "Show Tabs" menu item, and make sure "Visualize" and "3D Tools" are selected (Figure A 1). Once selected, the correlating tabs will appear in the middle of the Ribbon.

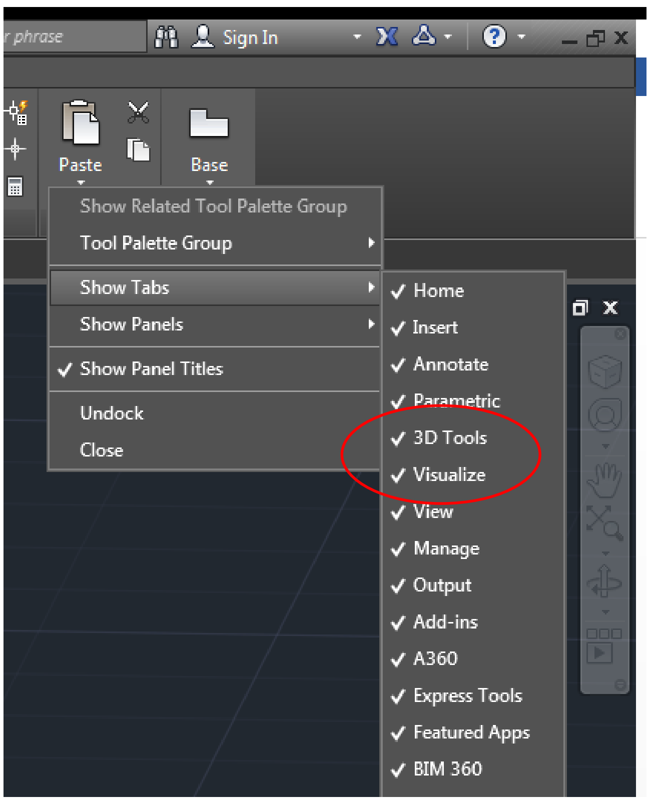


Figure A 1 - Initializing the AutoCAD "3D Tools" and "Visualize" tabs.

AutoCAD is a drafting program first, and a modeling program second, unlike some of the newer Autodesk software (e.g. Revit or Inventor). While drawing in 3D is possible, the recommended best practice is to draw in 2D then use the 3D tools to transform the object into a 3D object.

For generating a shell or solid structural element, start by drawing outlines of the structure that can be connected to make the final shape. In the example below, the outline of the arch was defined every 10-feet in elevation. Each outline was drawn as a 2D element, then the "Loft" command was used to connect the elements into a 3D shell element, called a "surface" in AutoCAD.

Step 1: Draw an outline of the Structure

Draw an outline of all of the changes in geometry in 2D space, then assign them to the correct elevation (this is your Z axis).

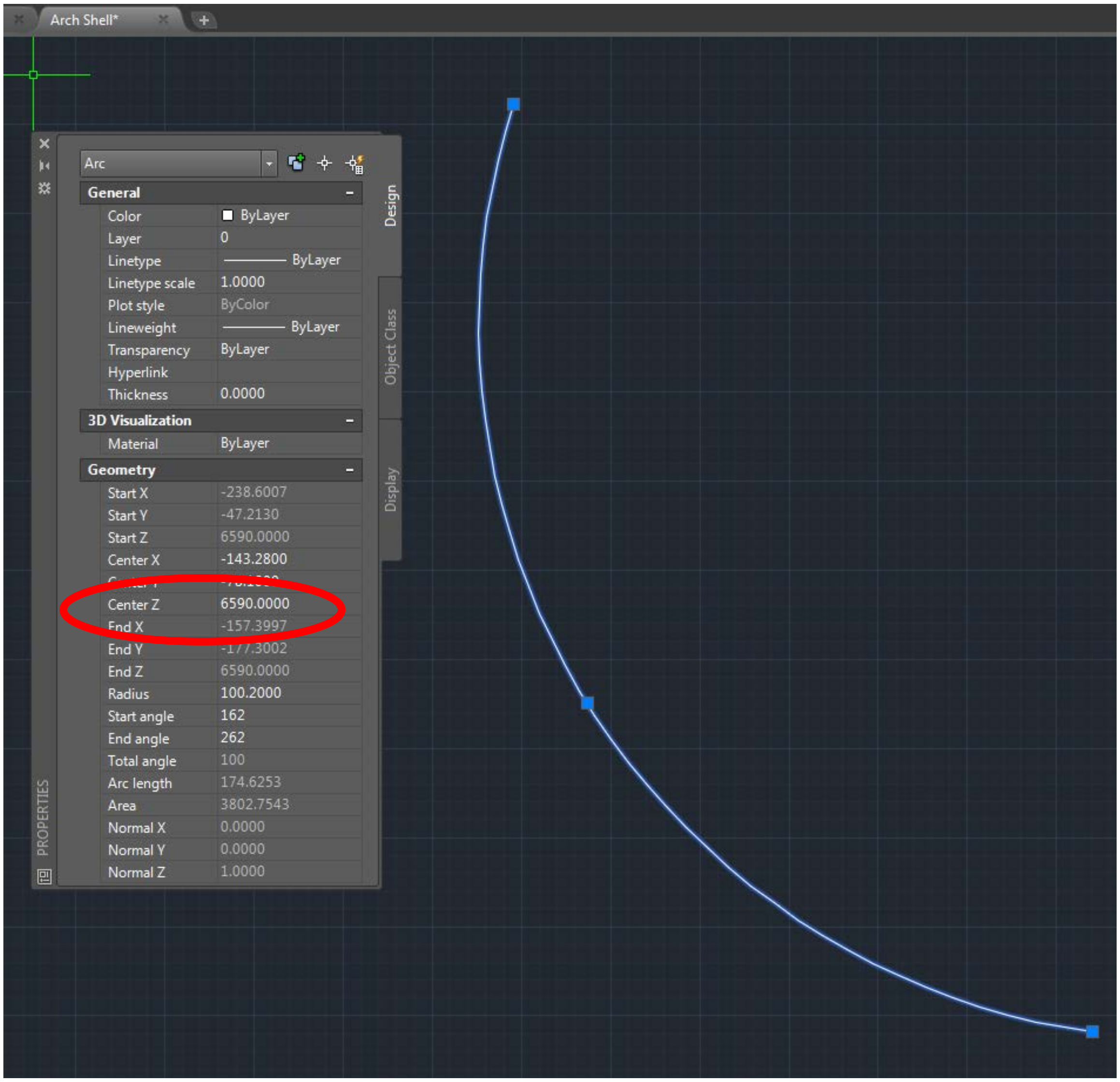


Figure A 2 - Outline element in AutoCAD.

In the example shown in Figure A 2, the object used is an "Arc" object, so setting the elevation is set by the "Center Z" point. Since the arc is a 2D object, the "Start Z" and "End Z" points will be at the same Z location as the Center Z.

Step 2: View all objects in 3D space

Once all of the outlines have been drawn in 2D and moved to the desired elevation in space, rotate the view using the "3DF" command or the Viewcube in the upper right corner to view the objects isometrically (Figure A 3).

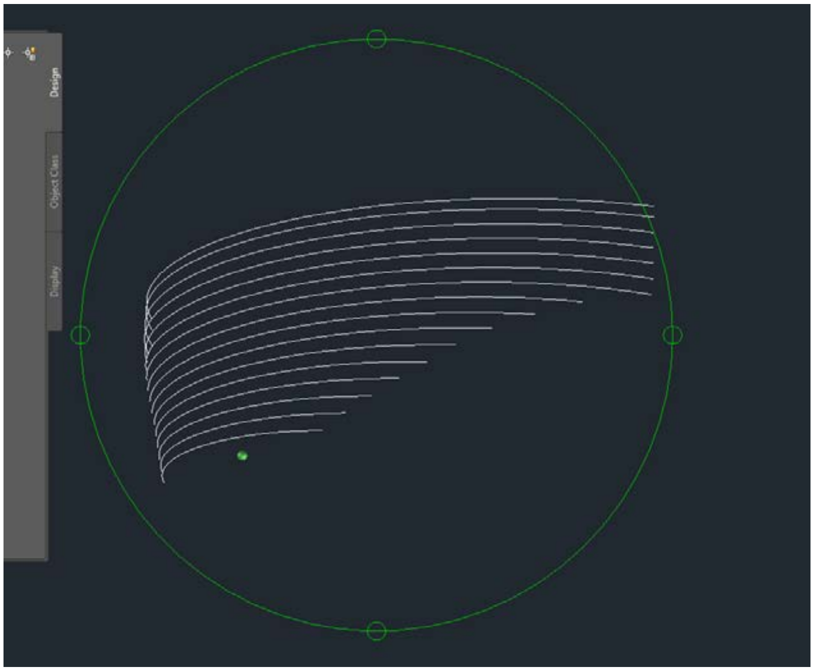


Figure A 3 - Isometric view of 2D objects defining arch geometry

Step 3: Use 3D Tools to transform into a 3D object

For this setup, with multiple cross sections, the use of the “Loft” command is ideal. If the structure is a similar shape, like a tower, and 1 cross section represents the section, use the “Extrude” command. If the cross section follows a curve, use the “Sweep” command. For all of these commands, AutoCAD prompts the user for the next step.

When using the Loft command, the first step is to select the “Loft” option from the “3D Tools” ribbon tab (Figure A 4), or type the command “Loft” into the Command Line. Following that, select the sections in the direction that the loft is proceeding (Figure A 5). After selecting all of the sections, hit the enter button to exit the selection then again to exit the command.

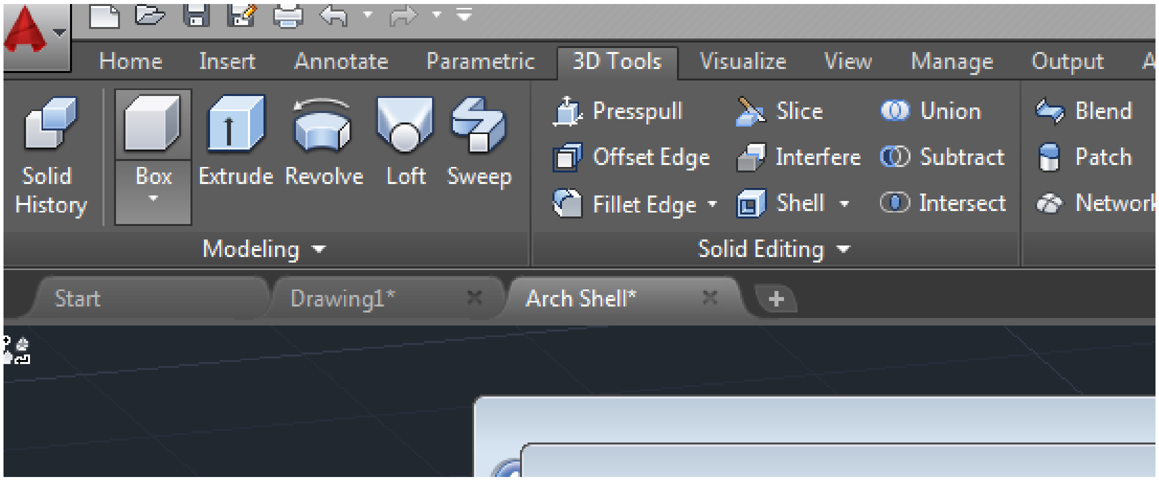


Figure A 4 - Partial view of "3D Tools" tab on the ribbon showing the basic modeling commands.

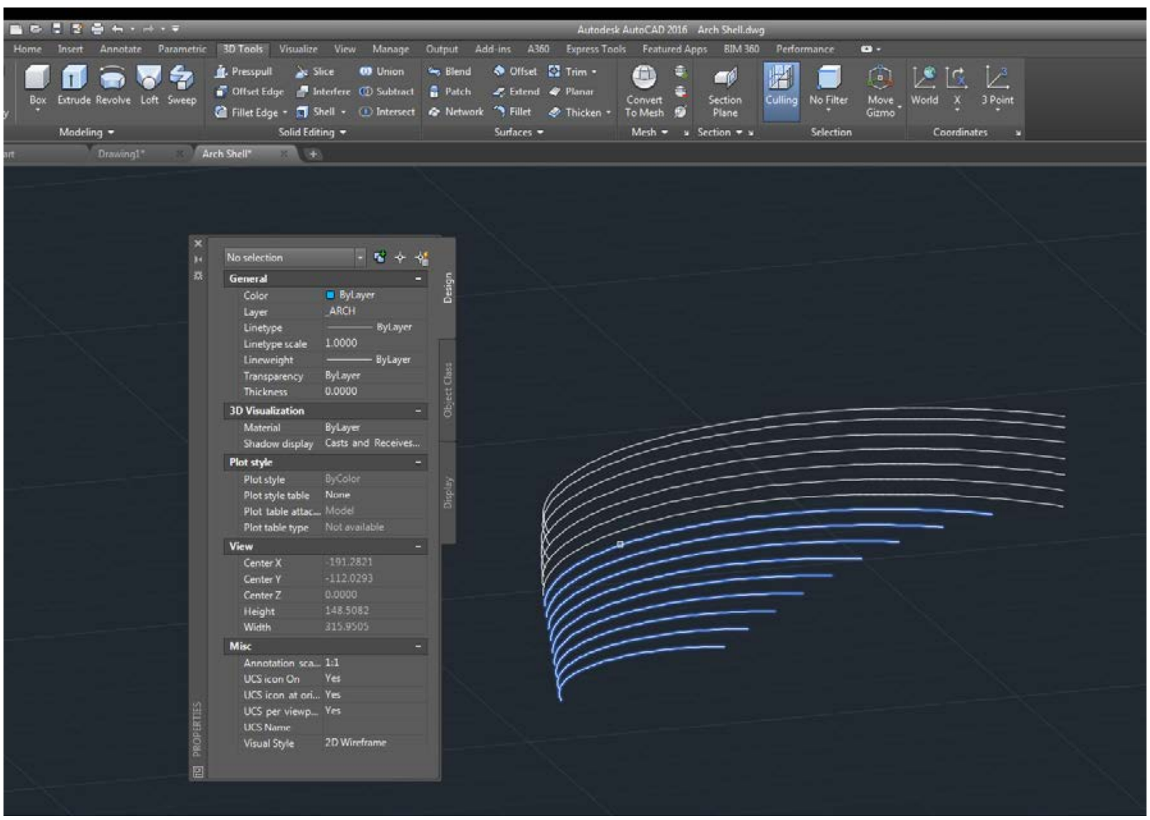


Figure A 5 - Selecting the 2D outlines in AutoCAD during the Loft command.

Once the 3D object is created, there are still options to edit the loft if it didn’t predict the loft correctly. Figure A 6 shows a loft using the “Ruled” option, meaning that a straight line is drawn from one section to the next. However, sometimes a smooth fit represents the object more accurately. To change how the loft is generated, select the 3D object, then select the down arrow next to the loft symbol, somewhere near the object in your model space. From here, you will see the different loft options (Figure A 7).

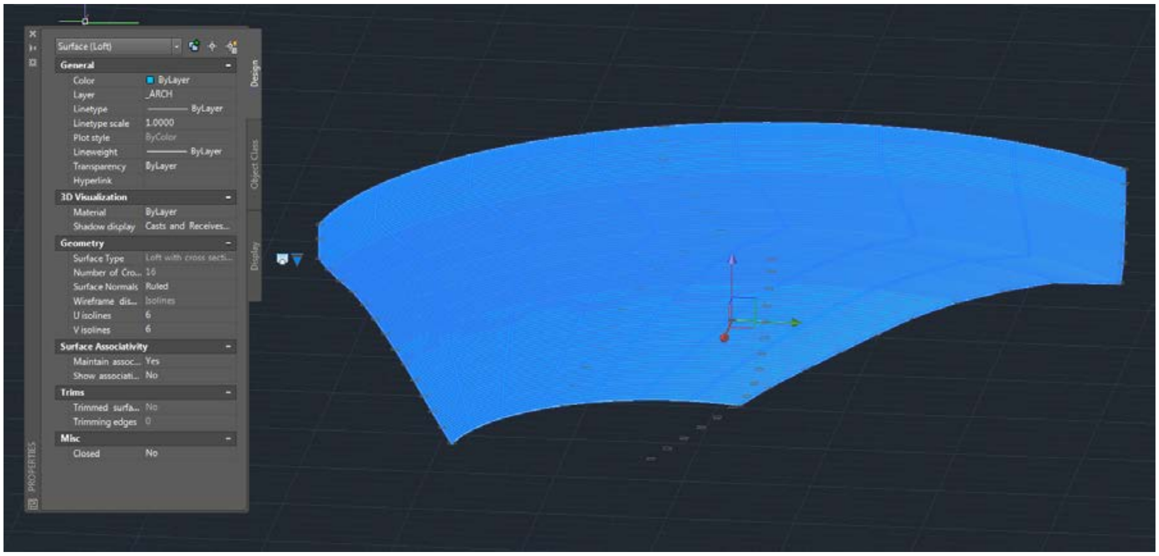


Figure A 6 - 3D Surface generated using the "Loft" command.

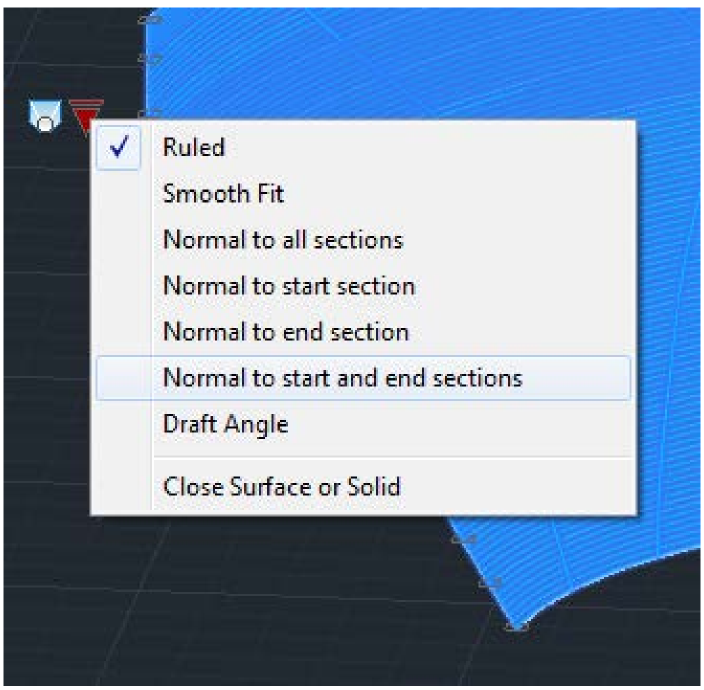


Figure A 7 - Loft command options after 3D object is created.

Step 4: Export the Model

Within AutoCAD, there are two exporting options that can be imported into LS-PrePost, the finite element software we are using: .stl and .igs/.iges (Figure A 8). Below is a brief description of the file types:

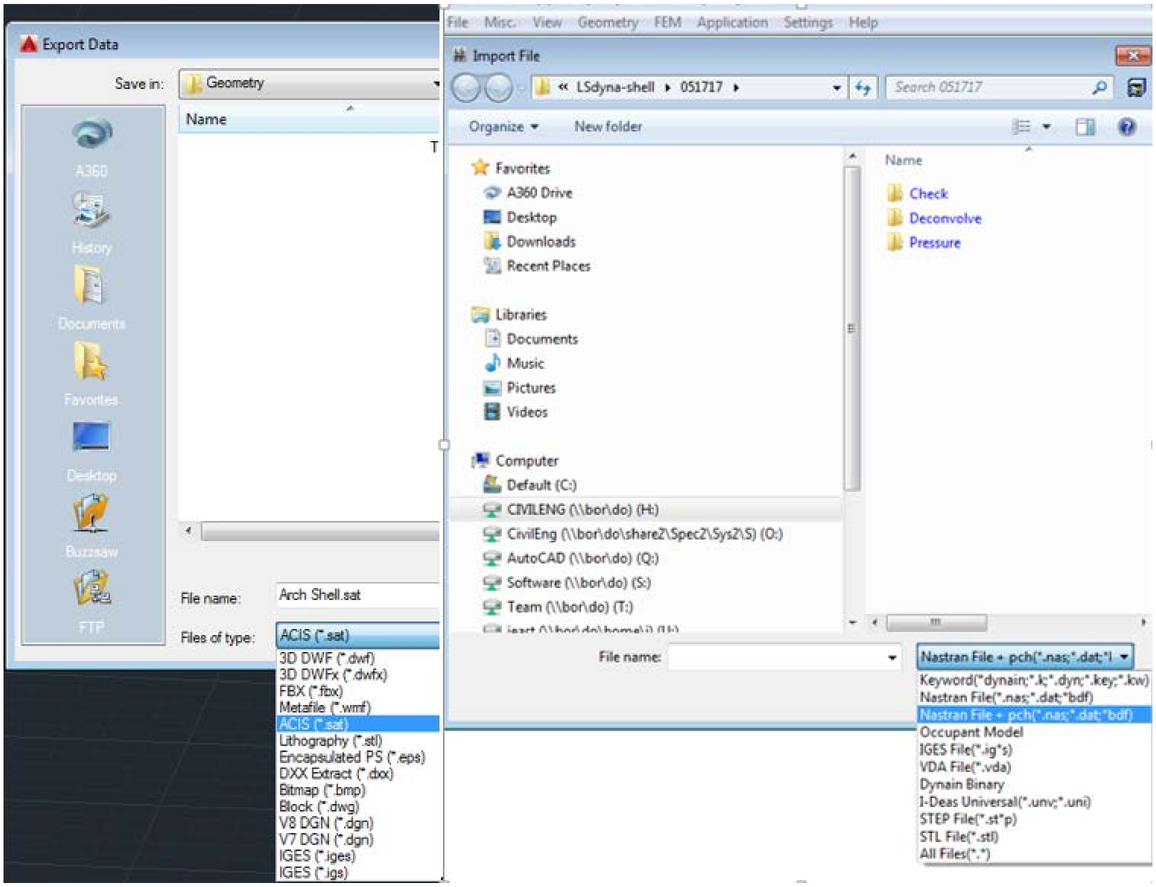


Figure A 8 - AutoCAD export options and LS-PrePost import options

- .stl: Stereolithography (stl) is a file format is a CAD format triangulating a drawing to represent various shapes. STL files do not describe properties such as layer or color, only the geometry [1].

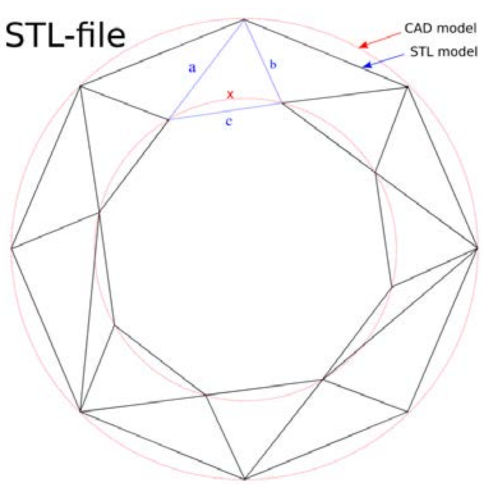


Figure A 9 - STL model vs CAD model [1]

- .igs/.iges: The Initial Graphics Exchange Specification (igs/iges) was designed as an independent way to exchange CAD information between programs from different developers. The format can include, among other things, 3D surfaces and solids. Additionally, the IGES file can support text, dimension lines, and other descriptive objects. IGES became an American National Standards Institute (ANSI) standard in 1980, helping keep it consistent between different software programs [2]. However, ISO has been superseded by STEP, which is not supported by AutoCAD [3].

Based on the descriptions, the .igs/.iges format was selected for export.

In AutoCAD, either type the command "igesexport", or: select the home icon, then "export", then "other formats" to open the export options. For file type, make sure .igs/.iges is selected. Specifying the new file name, hit enter, then select the object to export. Your export will work in the background.

Step 5: Importing into PrePost

In LS-PrePost, type "ctrl G" or select the "File" drop down menu, then "Import" then "IGES". At that point, the IGES import options will popup (Figure A 10). Typically, leave all of the options the same unless you want to change the units.

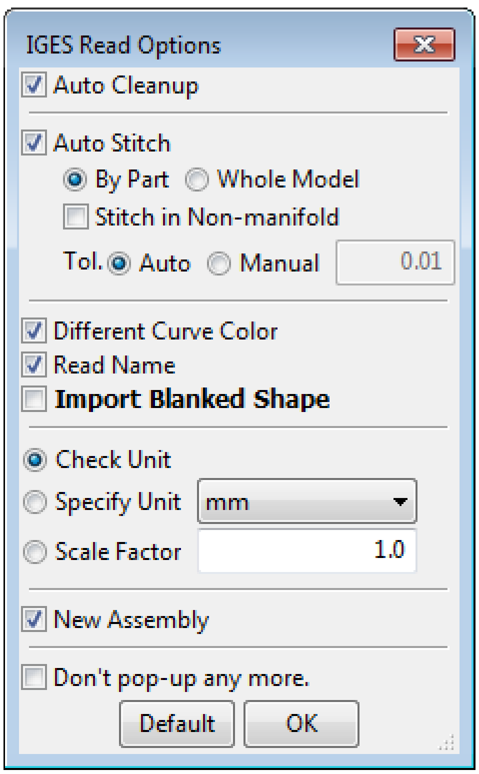


Figure A 10 - IGES Import Options.

*Note: Be sure to measure the model in LS-PrePost or check node locations to ensure the model is in the correct units.

You now have a working surface in LS-PrePost, However, it is not a working FE mesh.

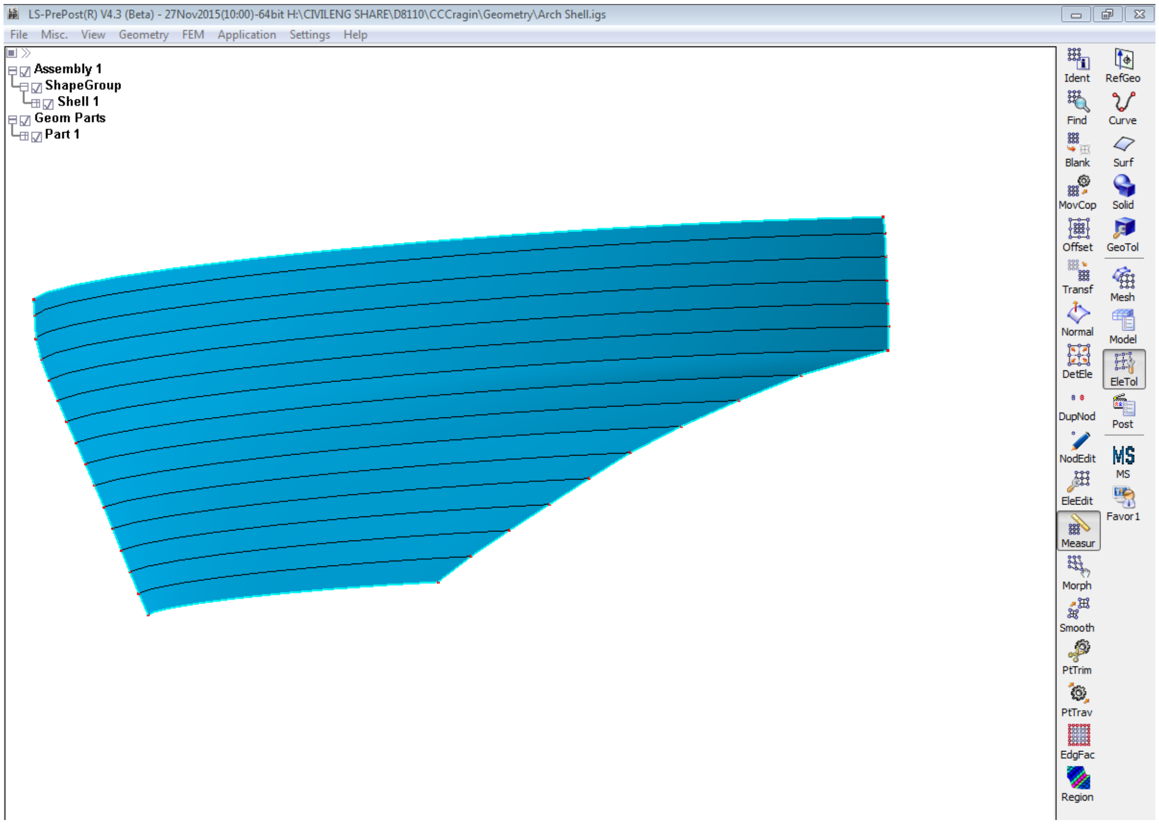


Figure A 11 - Shell model in LS-PrePost.

Now that the model is in LS-PrePost, you can edit it as desired for the FEA. This is the end of this tutorial.

ATTACHMENT B

AutoCAD Solid Model Development (SWEEP)

AutoCAD solid modeling is very similar to surface modeling. However, closed objects are required to make a solid, which is not required for a surface. This tutorial will demonstrate building a 3D solid model of a simplified hinge abutment and base block for an arch dam. This is more advanced than the surface model of the arch dam, but they do not need to be completed in any order as they are independent.

NOTE – For advanced models 3D CAD programs are recommended (e.g. Autodesk Inventor or Revit). An example of an advanced model would be something that changes in all three directions.

Step 1: Draw centerline of hinge abutment and base block.

The base block and hinge abutment are detailed in Figure B 1 and Figure B 2.

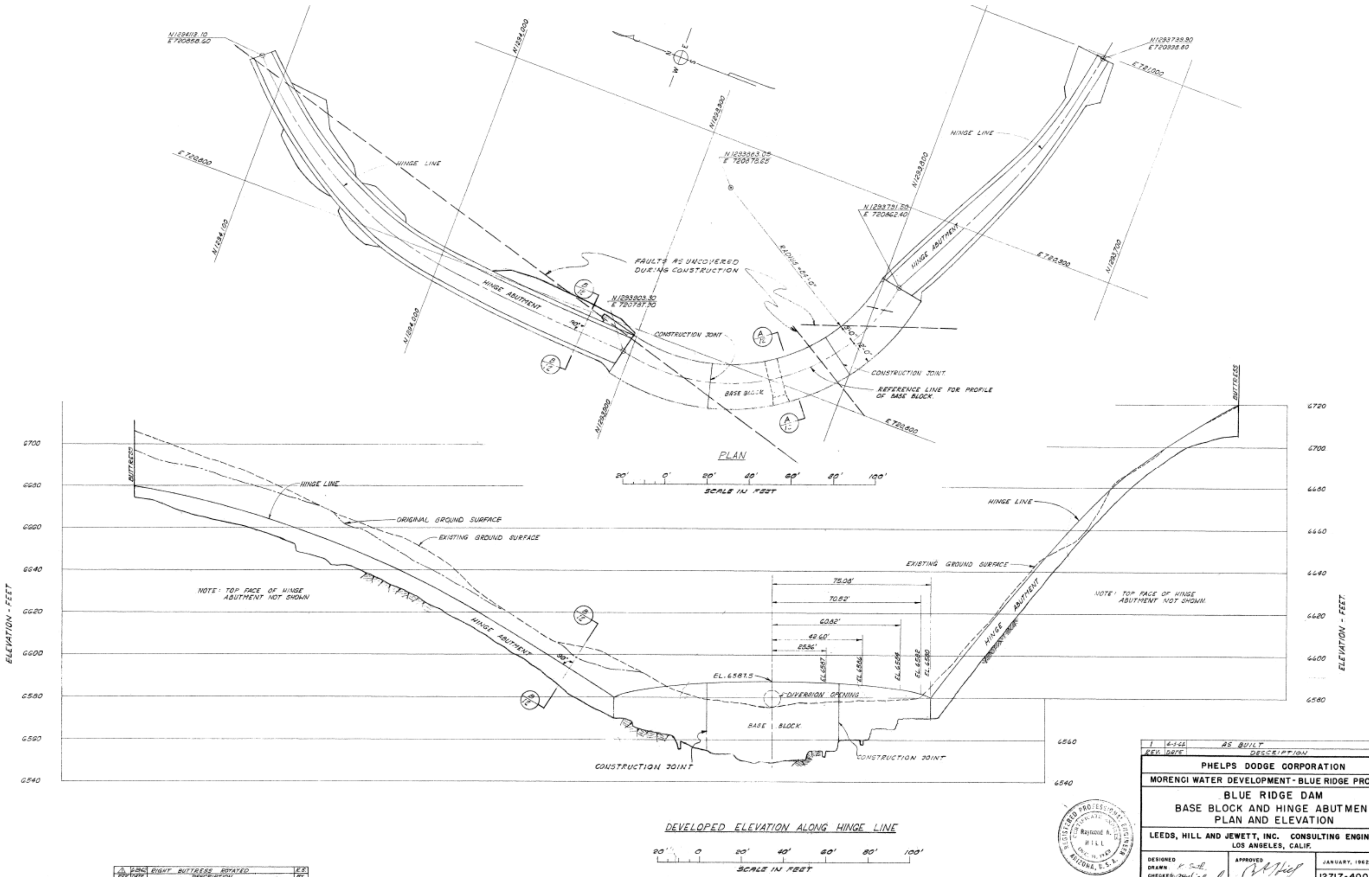


Figure B 1 - Plan and Elevation of hinge abutment and base block.

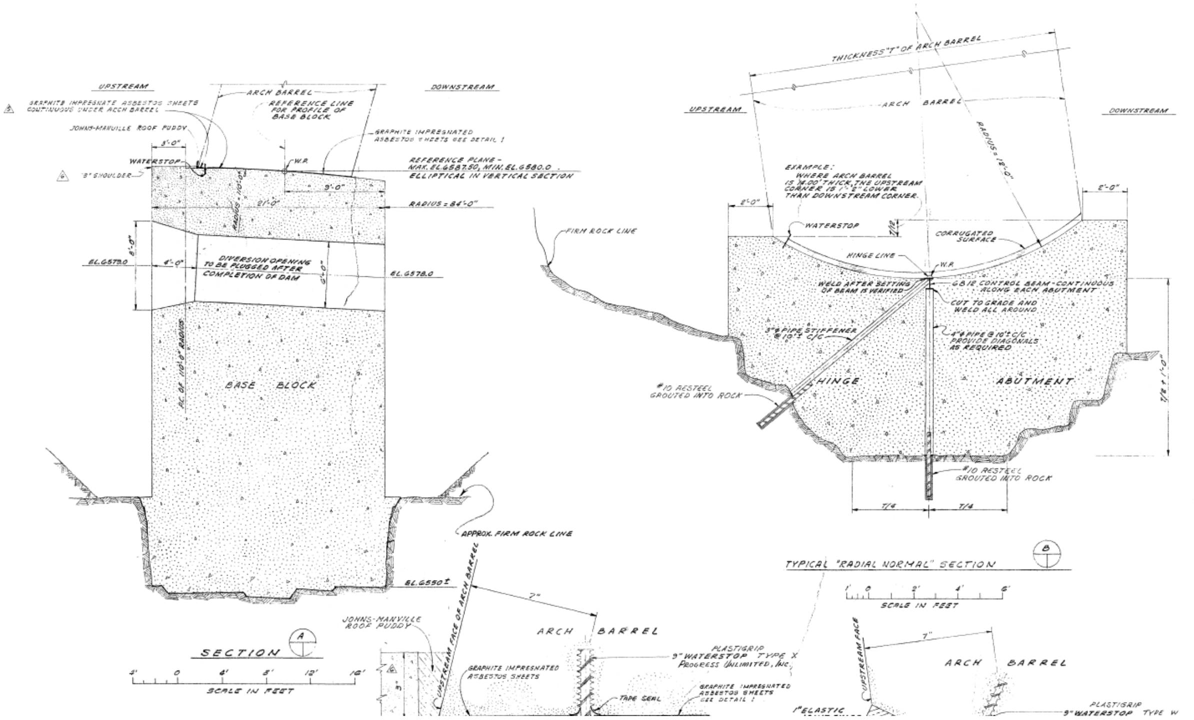


Figure B 2 - Critical sections of base block and hinge abutment.

The centerline of the base block and hinge abutment correlates to the endpoints of the horizontal arches as shown on Figure B 4. The endpoints of the arches and the thickness of the arches are given on that drawings. To easily generate the centerline, which will also be the line that profiles are “swept” along, the northing and eastings were typed into excel [4]. The AutoCAD command may be generated in excel and pasted into AutoCAD to expedite the process of typing each point into AutoCAD. Figure B 3 shows the AutoCAD generated abutments and Table B 1 shows the northing and easting layout.

Note: If the sweep path is a polyline or 3D polyline, the 3D object will be blocky with rough transitions from node to node (Figure B 3). To convert the polyline or 3D polyline to a spline, use the “PEDIT” command and convert it to a spline curve with the “S” option.

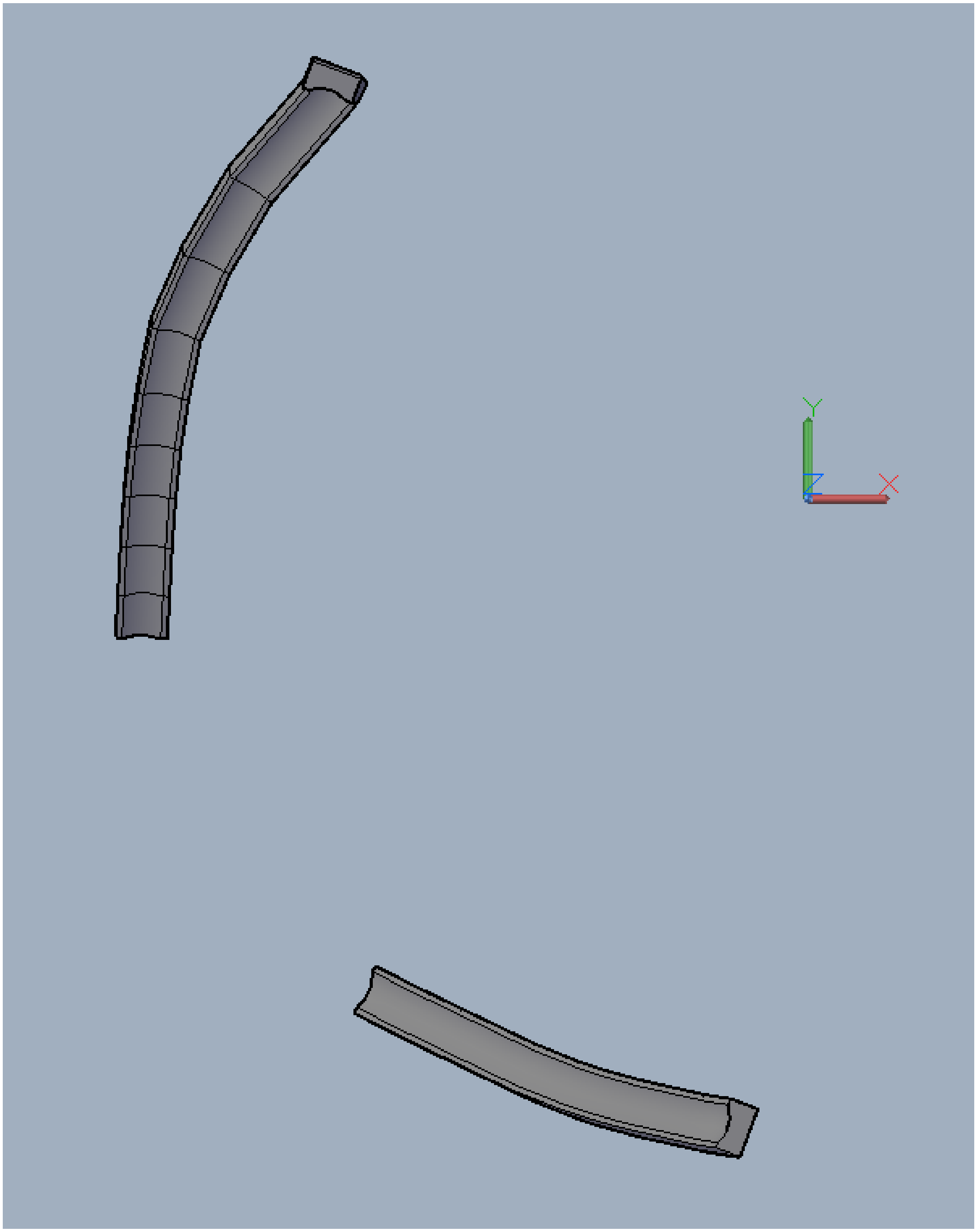


Figure B 3 - Example of a spline path (lower) vs a polyline path (upper).

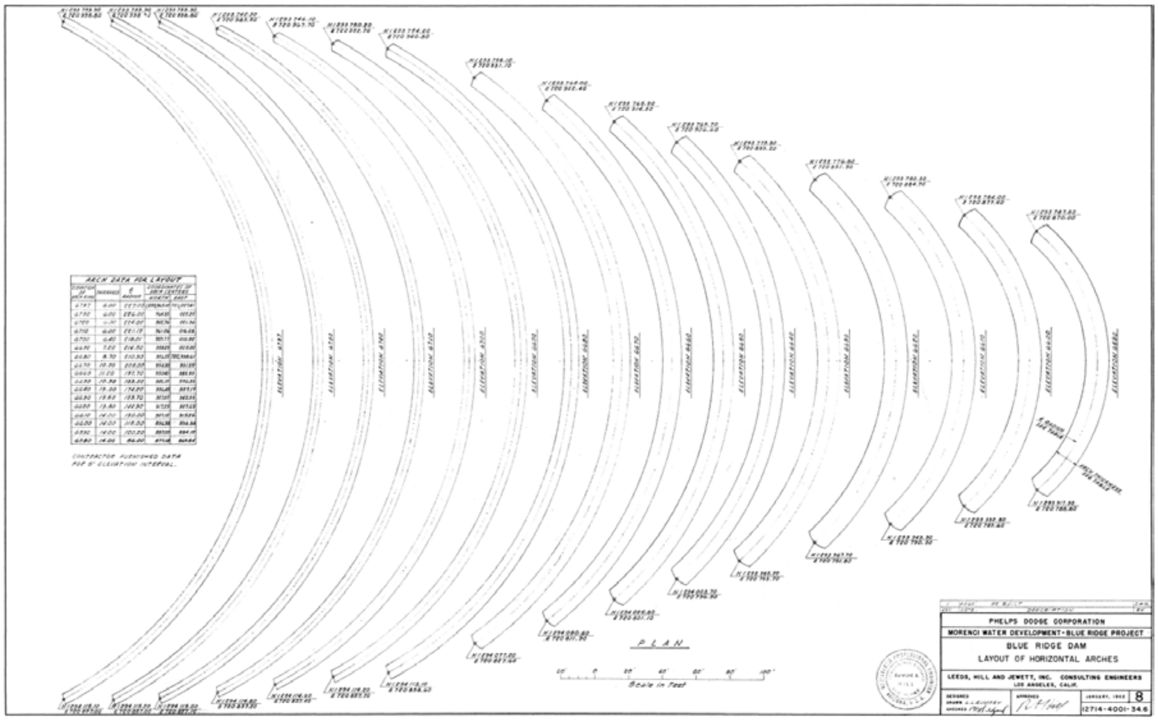


Figure B 4 - Layout of horizontal arches.

Use Excel and AutoCAD to easily input points

Utilizing Excel, or any other spreadsheet program, to organize critical points and input data into AutoCAD is important to expedite building the model. If a line needs to be created through a list of data points, such as the centerline for the base block and hinge abutment, the points can be manipulated in excel to draw the line using copy/paste commands.

The data need to be organized so each data point is in one cell in the order it will be entered into the spreadsheet (i.e. for the 3D Polyline command, the input is “X,Y,Z”, therefore each cell should have the X value, a comma, Y value, a comma, and the Z value, so it appears exactly as it would be typed). This saves time because typing all of these in accurately would take a long time and be prone to significant errors.

Table B 1 - Example of Excel table for AutoCAD input.

Left Abutment			For AutoCAD
Easting	Northing	Elev	(x,y,z)
227.0233	48.6448	6737.0	227.02328,48.644798,6737
227.0883	48.6258	6729.9	227.08833,48.625839,6729.8604
226.9523	48.6384	6722.7	226.95233,48.638424,6722.7212
226.7891	48.6590	6715.6	226.78911,48.659031,6715.583
226.6360	48.6694	6708.4	226.63596,48.669388,6708.4443
226.4308	48.6719	6701.3	226.43077,48.671932,6701.3066
226.1076	48.6880	6694.2	226.10759,48.687992,6694.1748
224.5013	48.7581	6680.0	224.50127,48.758137,6680
231.8508	68.3601	6660.0	231.85083,68.360123,6660
234.3090	74.9876	6650.0	234.30899,74.9875733,6650
234.1141	80.5099	6640.0	234.11406,80.509918,6640
234.5987	85.0114	6630.0	234.59865,85.011429,6630
235.6243	89.3678	6620.0	235.62434,89.36776,6620
237.3971	93.6710	6610.0	237.39711,93.671028,6610
239.9935	97.7347	6600.0	239.99352,97.734741,6600
243.2336	101.1894	6590.0	243.23358,101.18936,6590

The Excel “Concatenate” command is used to combine strings together within a cell. In this example, assuming the first row of text is row 3 and column A, the first Concatenate command is =CONCATENATE(“A3, “,” , B3 , “,” , C3). The comma without quotations (,) is the separator for each input, while the comma within quotation (“,”) is a text string that will be shown in the cell.

To enter this into AutoCAD: copy the column of inputs in Excel, then in AutoCAD start the command, then click within the Command Line (where commands are typed), and enter Ctrl V to paste the values. Hit the “enter” button to close the command. Figure B 5 shows the 3D polyline (“3Dpoly”) generated by the table above.

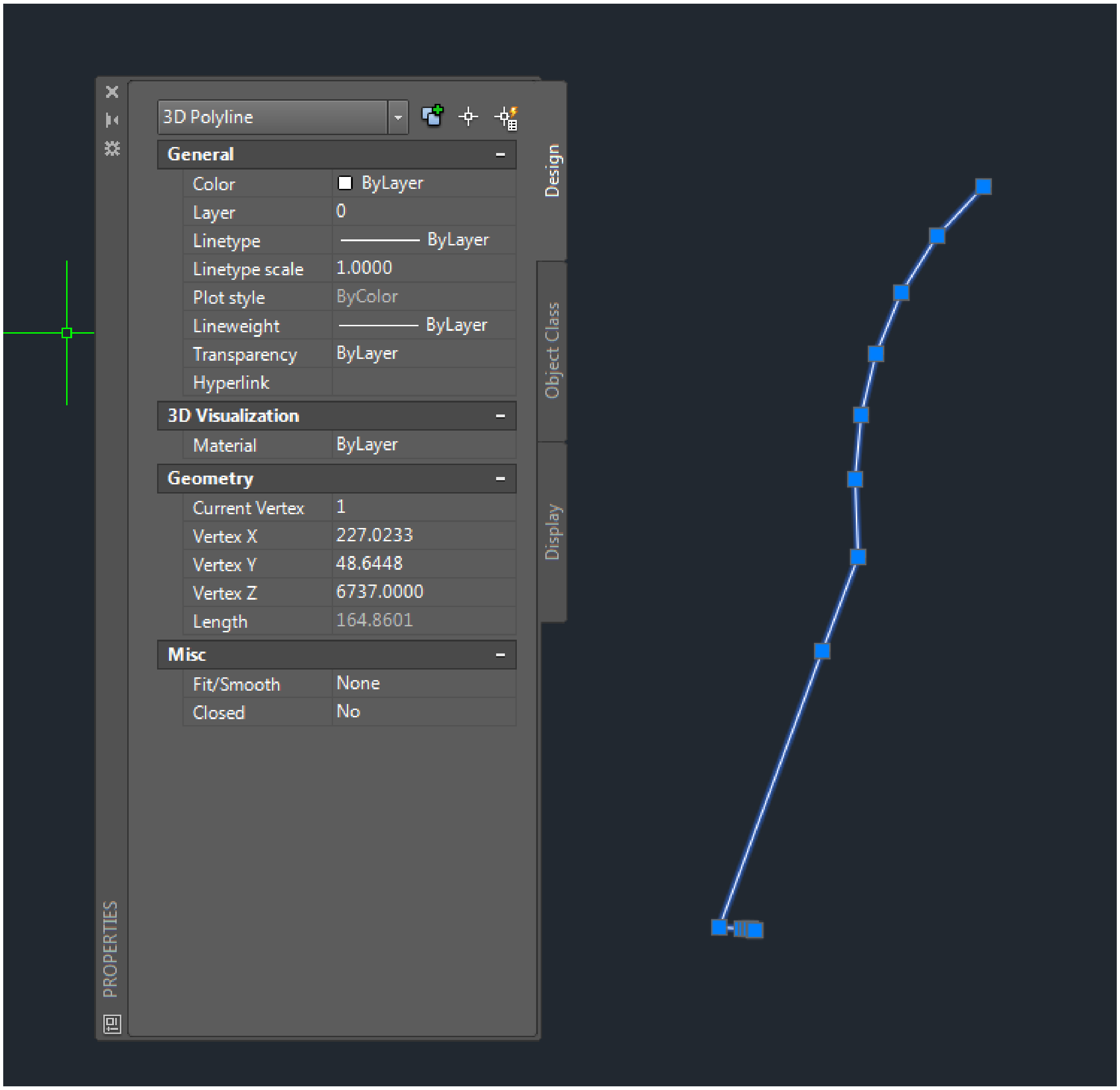


Figure B 5 - 3D Polyline generated using Excel table of values.

Typically, the line drawn will not be shown on the screen. Therefore, enter the Zoom command ("Z") then enter the Extents options ("E"). That will show the extents of what is drawn in model space.

Step 2: Draw section profiles of the model along the centerline

This step is similar to Step 1 for “AutoCAD Shell Model Development.” The idea is to draw the significant geometry profiles to “sweep” along the centerline. Figure B 6 shows the hinge abutment section drawn to be extruded along the centerline path[1].

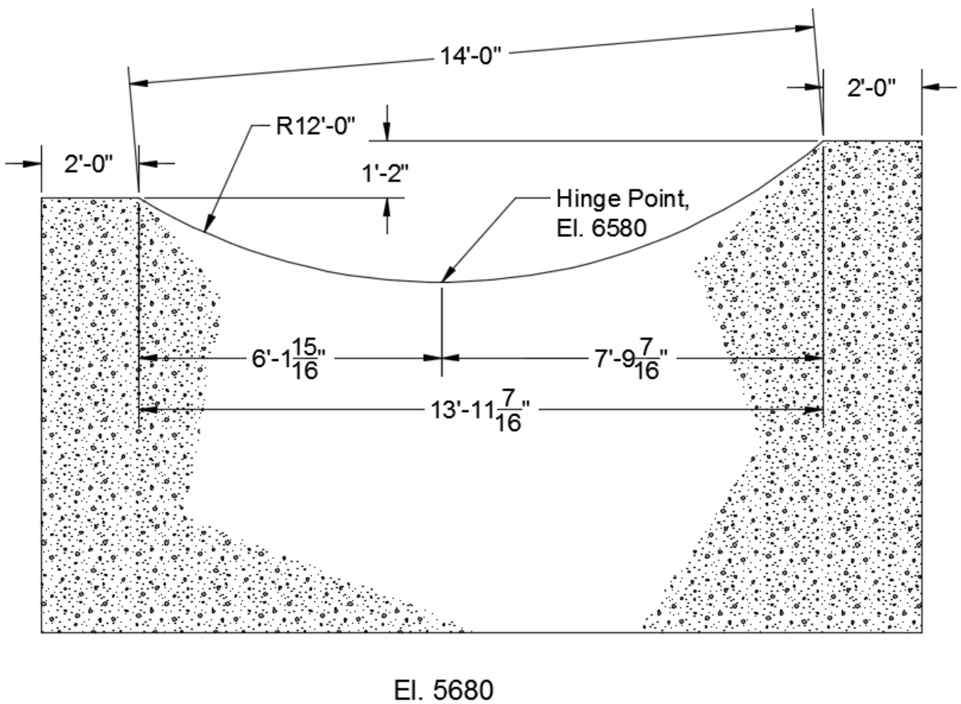


Figure B 6 - Section through hinge abutment for AutoCAD 3D model

The section through the hinge abutment will be swept along the centerline based on the hinge point in the profile.

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[1] A critical profile was determined to be swept along the abutment. Because the hinge abutment is parametric based on the thickness of the arch at a certain elevation, it is too complex for AutoCAD 3D modeling tools. However, with engineering judgement structures can sometimes be simplified to be generated more simply without significant loss in structural characteristics.

Step 3: "Sweep" the Section along the Centerline

This is the step where the actual model is being built. The "sweep" command is similar to "extrude", except the shape is pulled along a line instead of perpendicular to the cross section.

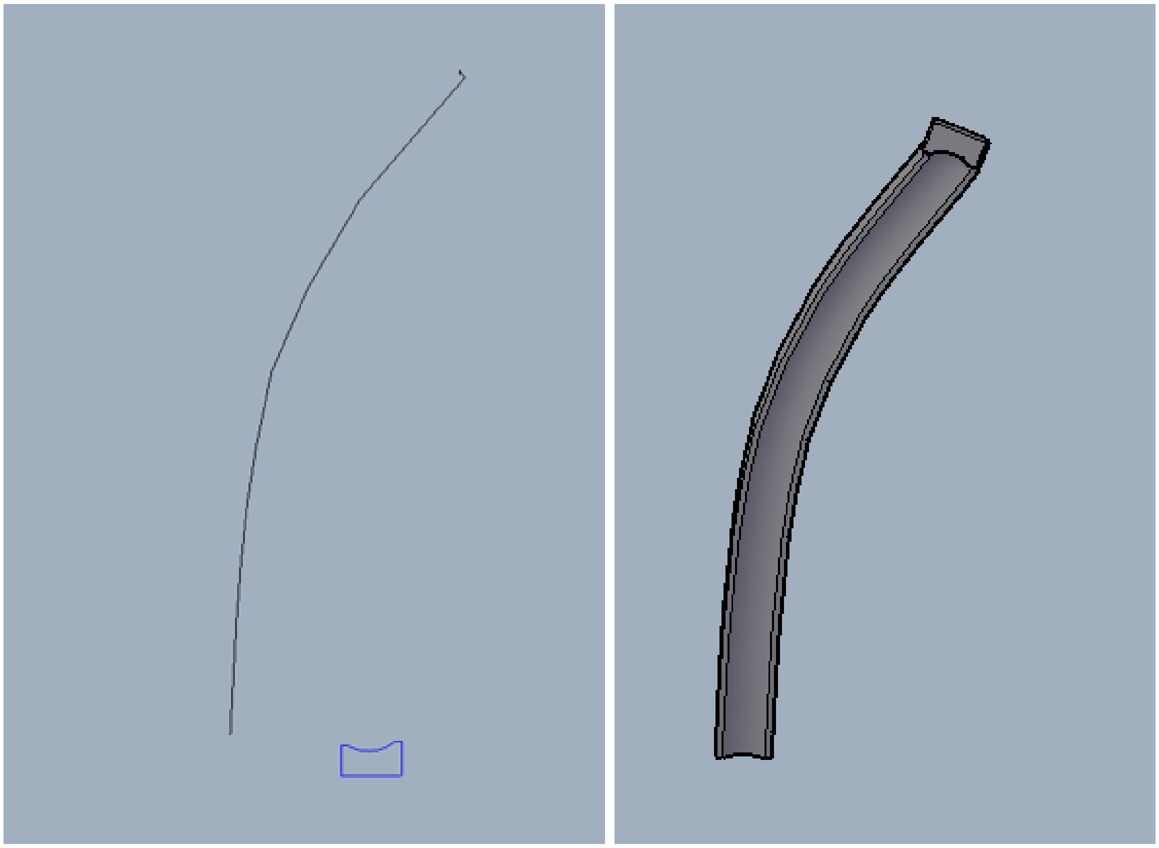

Figure B 7 - Section and guideline for sweep command on the left, 3D model on the right

With the guideline (centerline in this case) and the section drawn, start the "Sweep" command by typing "Sweep" in the command line or selecting the "Sweep" button in the "3D Tools" tab in the ribbon and follow the prompts:

1. Select objects to sweep – Select the section object
2. Select the sweep path or [Alignment, Base point, Scale, Twist] –
    a. At this point you can enter other options to control the sweep. The alternative options are "Alignment, Base point, Scale, Twist". Select "Base Point" or type B and press enter.
        i. In the "Base Point" option, set the point that will follow the centerline. In this case, select the hinge point.
        ii. After this, the command returns to the "Select the sweep path or [Alignment, Base point, Scale, Twist]
            1. Alignment allows the user to orient the section other than perpendicular to the path
            2. Scale allows the user to scale the section (either directly or with an equation)

    3. Twist sets a twist angle of the section, such as a corkscrew

  b. Select the sweep path to finish the 3D model

Note: Sweep works left to right, so if the shape is mirrored, mirror your section, or rotate your section, as required.

You may now export the model into PrePost using Steps 4 and 5 from the “AutoCAD Shell Model Development” Attachment tutorial.

ATTACHMENT C

AutoCAD Civil 3D Surface Model Development

Note: The following tutorial was designed assuming that only hard copies of drawings were available and that points were going to be exported to Truegrid. Although this tutorial references Truegrid, the exact same procedure, except formatting

Creating a Surface in Civil 3D to Export Points or Contours

This approach is going to be based on using existing drawings and tracing lines as contours to use in surface creation. The surface in Civil 3D may be constructed in multiple ways, such as: using USGS survey data, ArcGIS Shape files, bathymetric survey data (reservoir floor), etc. Tracing contours will likely be the most common, therefore it was used here. Additionally, if a dwg file already has polylines with elevation data (contours), the tracing step may be skipped.

Step 1 – Insert image or pdf file

First, find a drawing that shows contours for the desired site. The drawing may be in any image format or a pdf. The easiest method for older drawings would be to find the drawing in eDraws; likely to be a tiff which can be directly uploaded and will be the cleanest scan.

In eDraws, search for the desired drawing and drag it to your working folder.

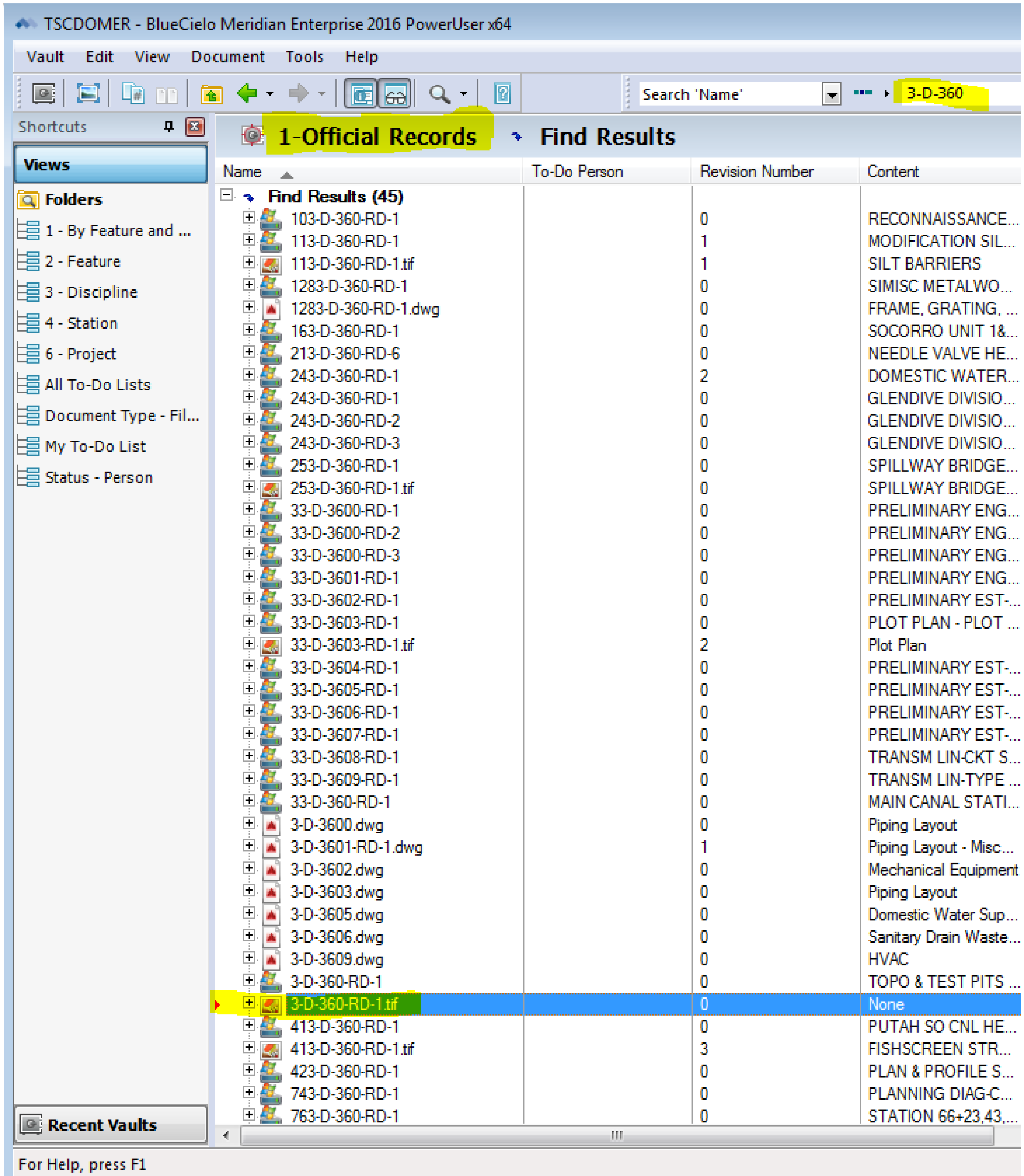

Figure C 1 - eDRAWS BC-Workspace drawing search.

Now, go into AutoCAD Civil 3D and type and execute the command ***xref*** which opens the External References window. Select the drop-down arrow next to the DWG image, then select attach image (or pdf if that is what you are working with).

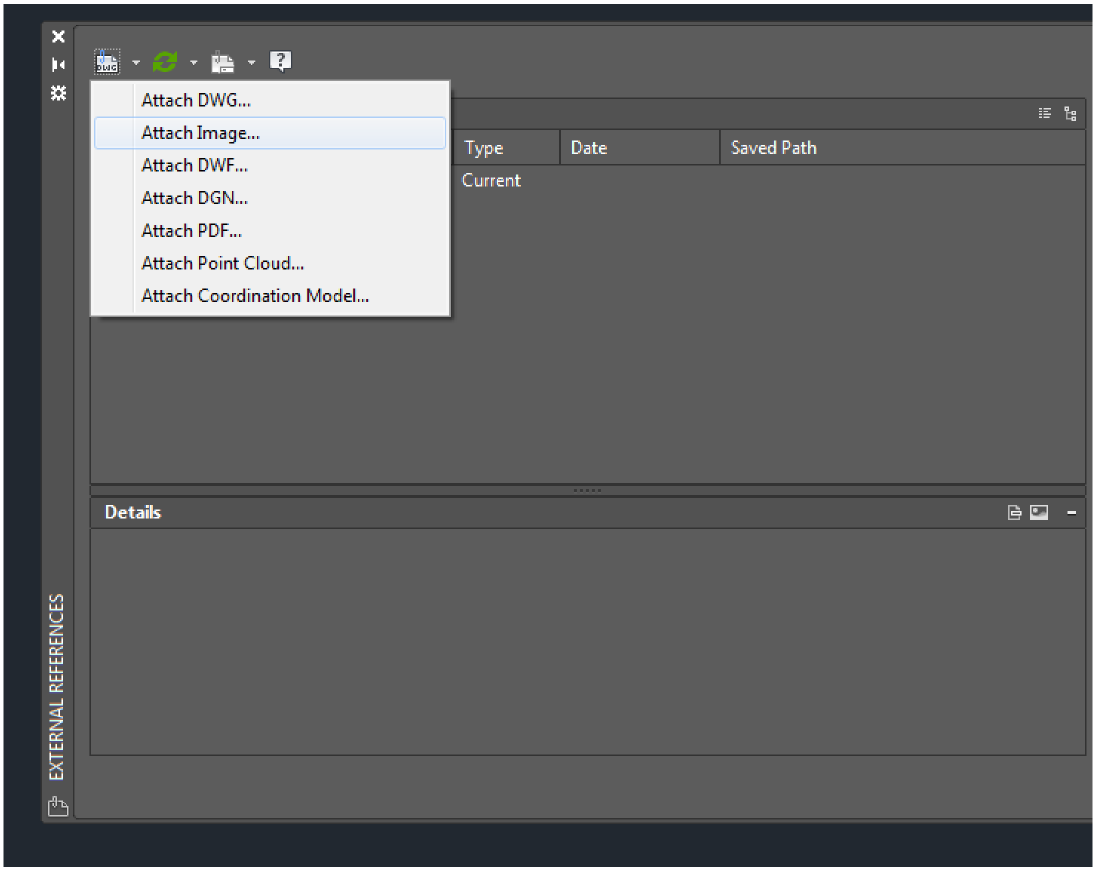

Figure C 2 - Uploading an external referance into AutoCAD.

This will open a windows explorer window from which you can select the desired reference file to be included. Once the file is selected, a new window will appear. Leave the defaults selected and select OK.

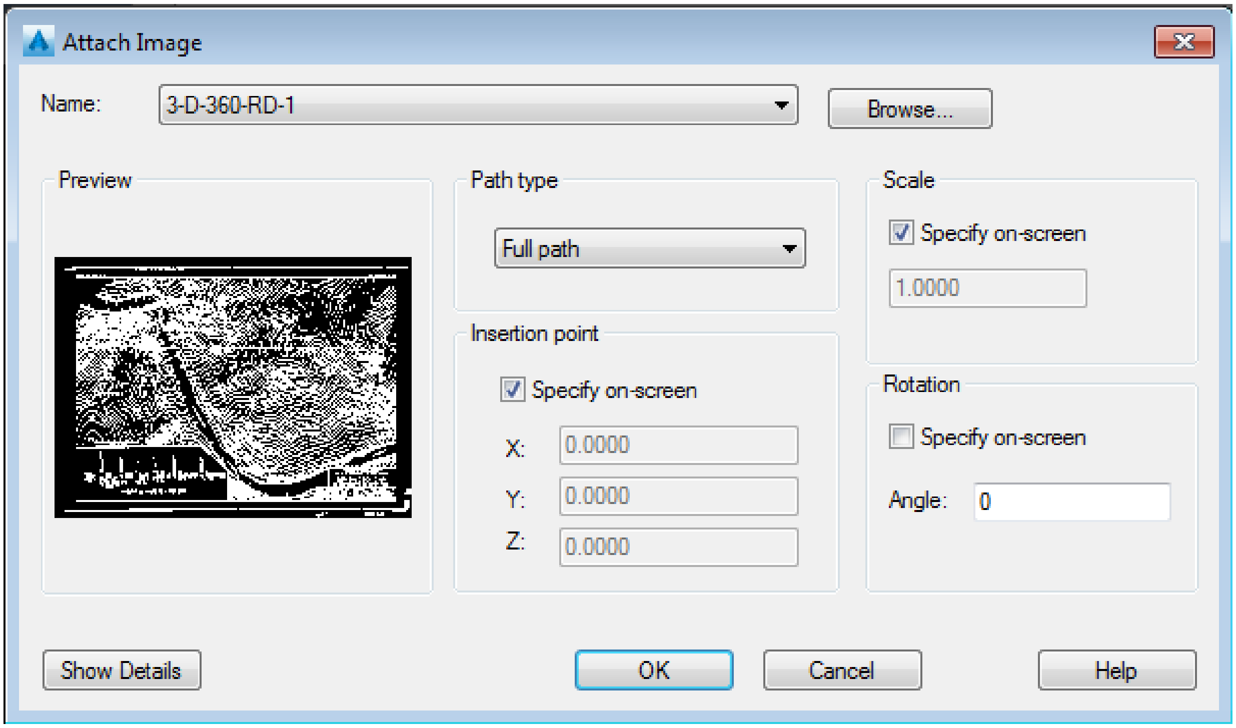

Figure C 3 - AutoCAD external reference attachment options.

Place the drawings anywhere, and specify any scaler – these will be corrected in the next step.

Step 2 – Scaling the Drawings and Moving it to the Desired Coordinate

Start by scaling the drawing so 1 drawing unit is equivalent to 1 foot. Using the ***PL*** command, draw a polyline the length of the scale bar. The polyline is in RED, below.

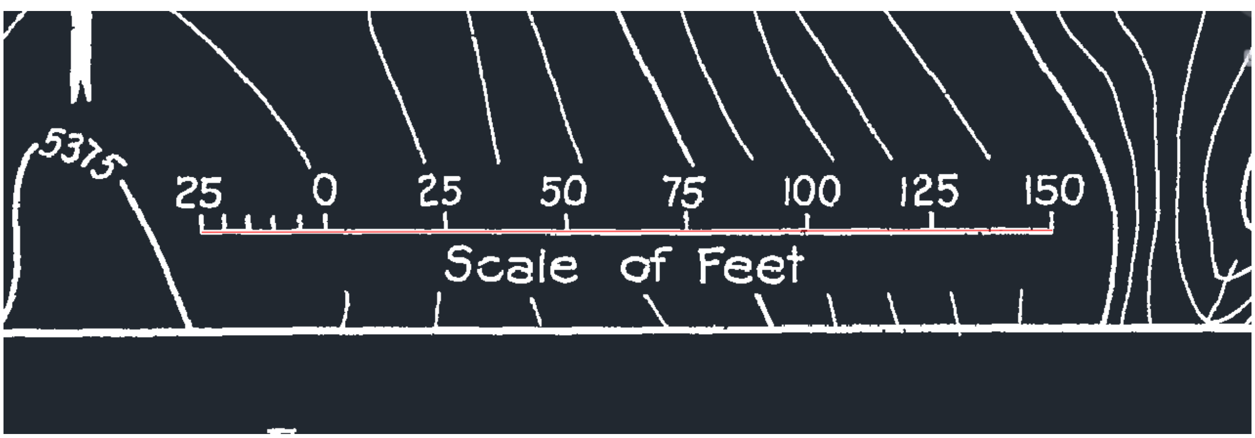

Figure C 4 - Drawing a line on top of a scale in AutoCAD.

Now, use the ***SC*** command and select both the image and the polyline to scale. Note, to grab the image, you will need to select the border. After the items to scale are selected, the command asks for a base point. Make sure your snaps are on (F3 button or the button shown below):

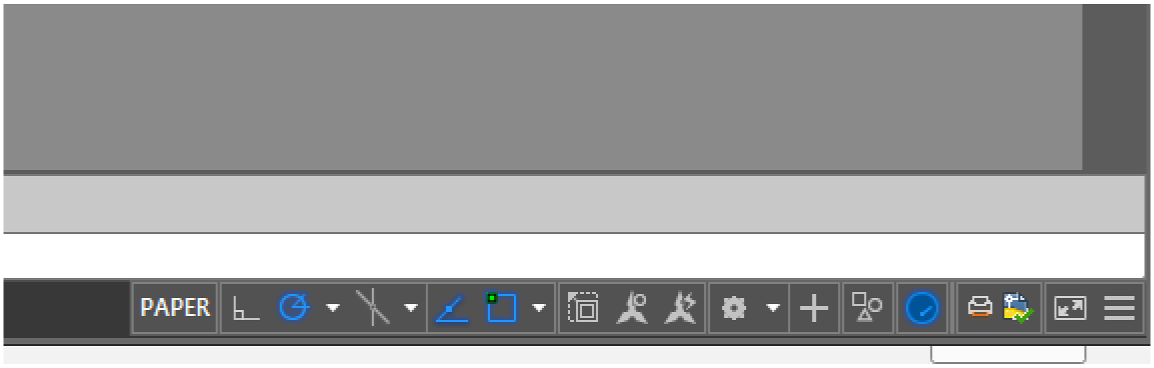

Figure C 5 - AutoCAD status bar with "snap" action highlighted.

Select the left end of your polyline as the basepoint. Then AutoCAD asks the user to specify a scale factor. Instead, enter ***R*** and hit enter – this is the alternate command option to scale based on a reference. It then asks for the scale length. First, select the left end of your polyline, then the right end. That is the reference length. Then it asks for the desired length- this is based on the scale. In the figure above, the length of the scale was 175 feet, so I entered 175 feet. Now, 1 drawing unit is equal to 1 foot in the image.

The next step is to orient the drawing so north is up if it isn't already. To do this, draw a line along the north arrow and a then a second vertical line crossing the first line.

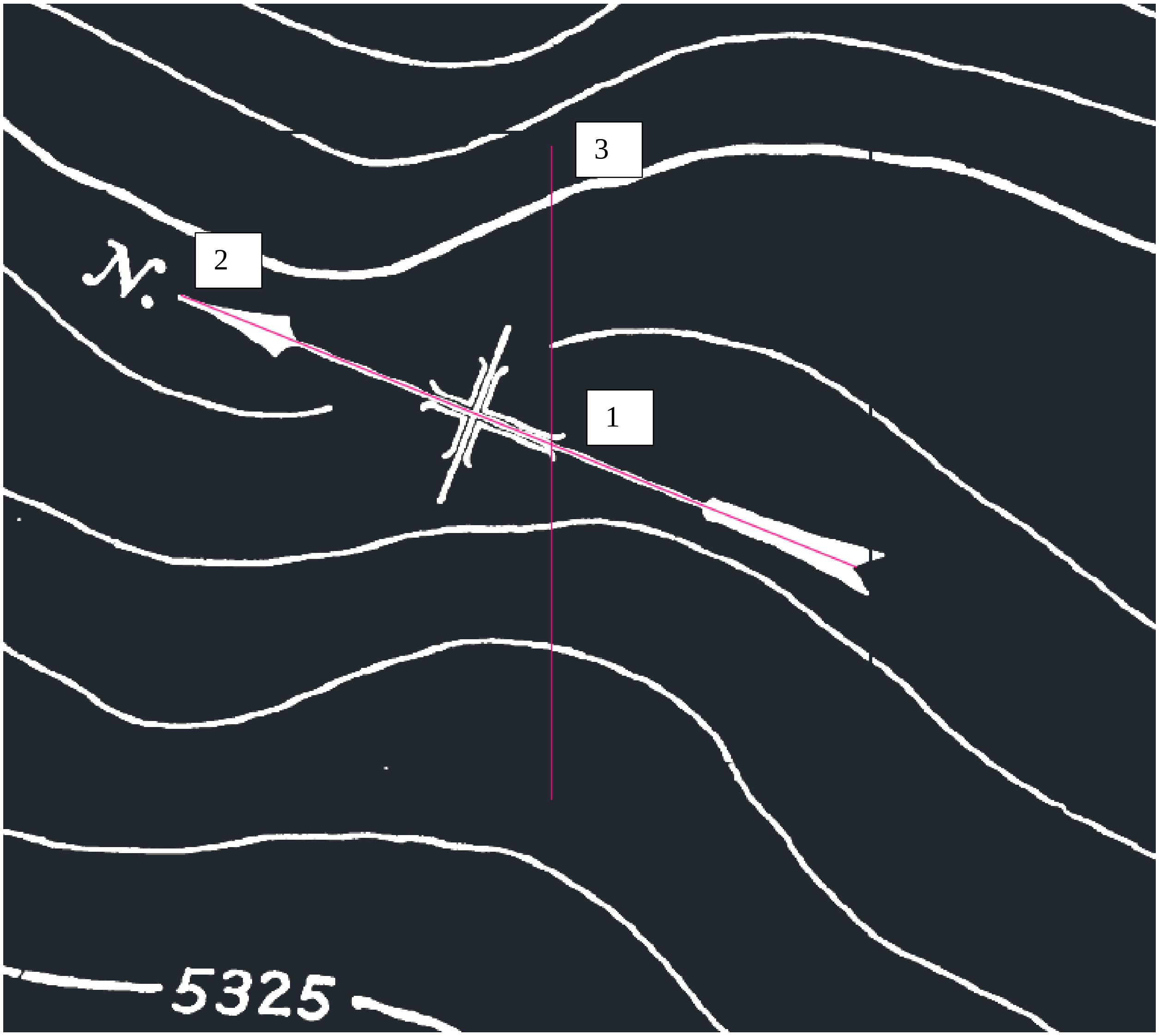

Figure C 6 - Orienting a figure based on the north arrow.

Select the line on the north arrow and the image, then initiate the rotate command (***RO***). It will ask for the base point, select the intersection of the two lines. Then it will ask for the rotation angle; again, enter ***R*** and hit enter to start the alternate reference option. Then select the intersection again, followed by the north end of the line along the north arrow, then select the top of the vertical line.

The final step is to move the drawing so it is in the right place in space. I prefer to use the northing and eastings shown on the drawing, but somehow it needs to be correlated to your Truegrid model. For the Truegrid model, DO NOT use northing and eastings – this will cause errors due to large numbers. I correct the coordinates after they are exported in excel.

To move the drawing to the correct northing easting, place a polyline at the northing easting mark on the drawing, then place a point or polyline at that location in model space. Then, select the drawings and initiate the move command (***m***), select the polyline point marking the N/E in the drawing as the basepoint, then select the actual northing easting as the location to move the drawing to.

Step 3 – Draw Contours

Trace the contours with polylines (***PL***) then enter the desired elevation in the Properties menu (***PR***). No need to be really exact, a 25-foot interval is typically acceptable, just be sure so capture major terrain features.

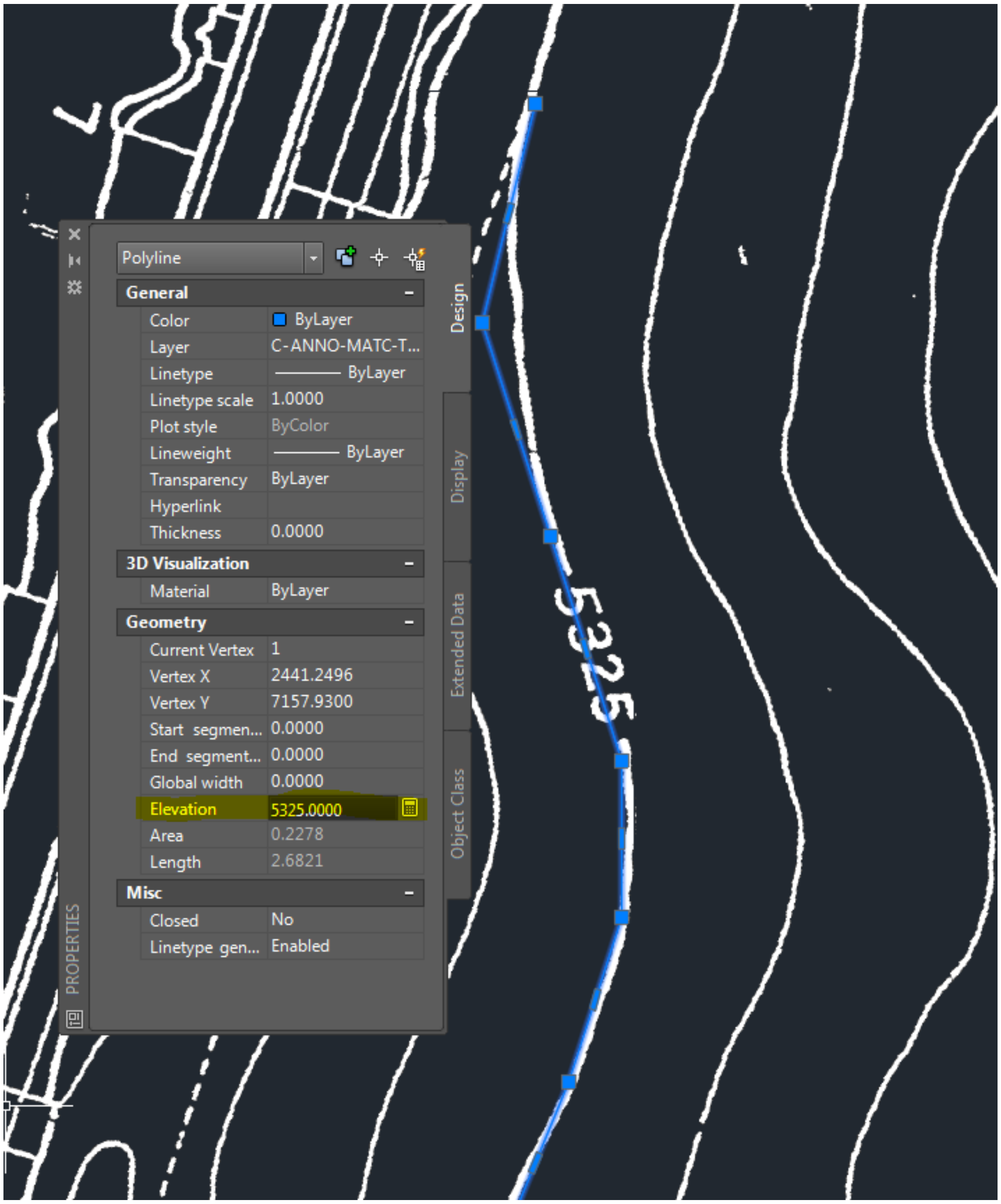

Figure C 7 - Example showing traced elevation lines in Civil 3D and inputting the elevation in the properties.

Step 4 – Create the Surface

Once all of the contours are in place, we now want to create the surface. Open the ToolSpace (***Toolspace***). In the Prospector tab of the tool space, right click "Surface" and select "Create Surface"

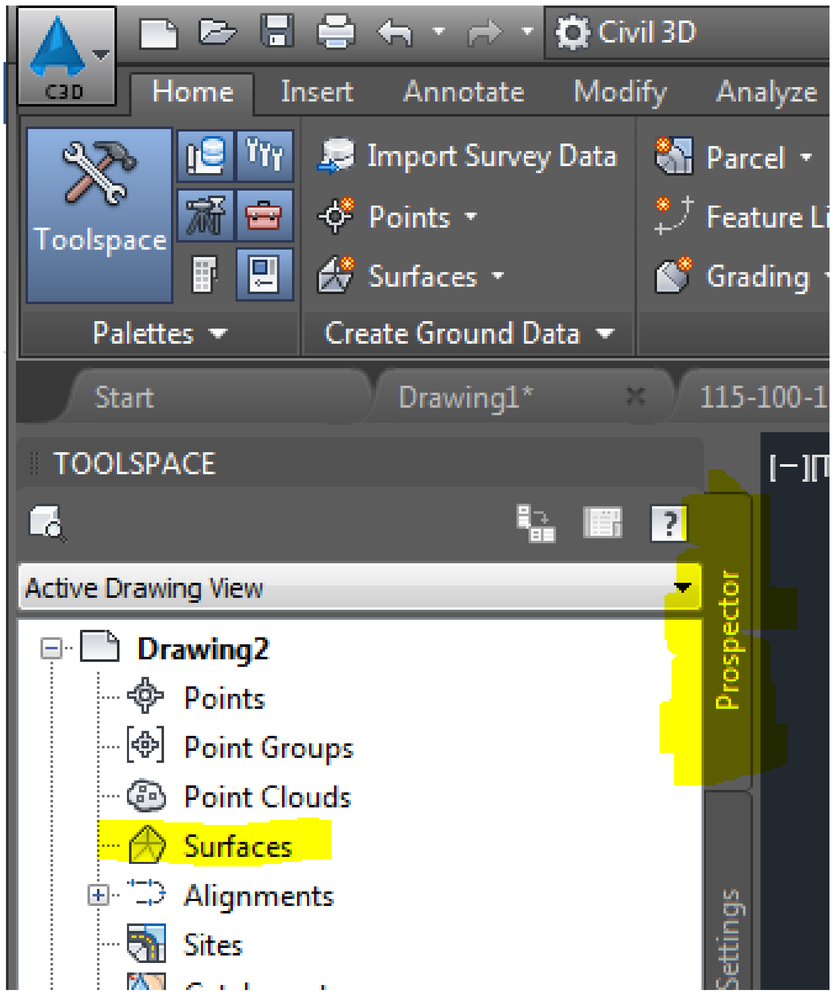

Figure C 8 - AutoCAD Civil 3D Toolspace Prospector tab.

This opens the surface dialog box where you create a surface placeholder – the name and surface type.

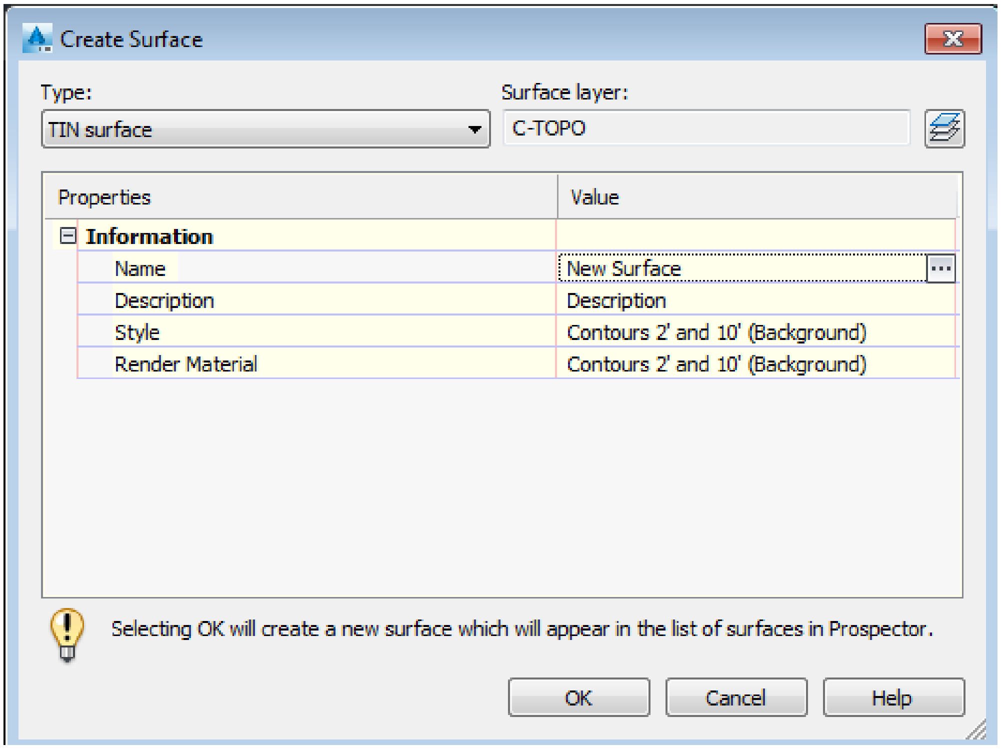

Figure C 9 - AutoCAD Civil 3D surface creation dialog box.

Select enter after you have entered the name. Be sure to use the TIN surface type. The Style is just how it is displayed, that doesn't impact the surface.

Now that the surface is created, there is a plus sign next to the Surfaces option in the ToolSpace. Open this, then open your new surface, then open the Definition tab – here we will define the surface.

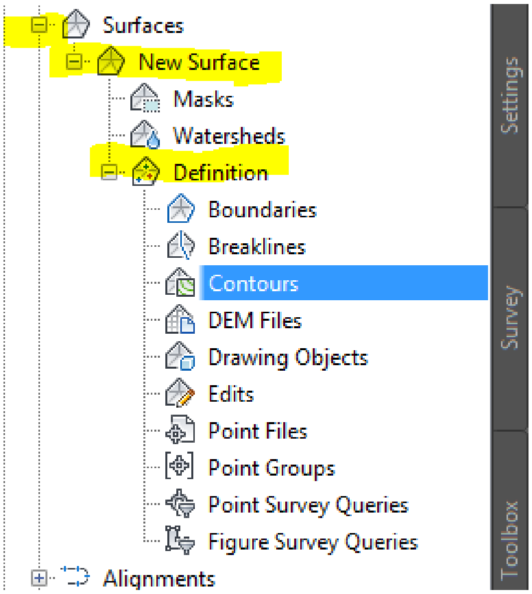

Figure C 10 - Toolspace Prospector tab surface definition options.

Select all of the contours in your drawing, then right click the “Contours” option in the definition and select “Add Contours”. A new dialog box will appear. Typically keep the defaults and select ok.

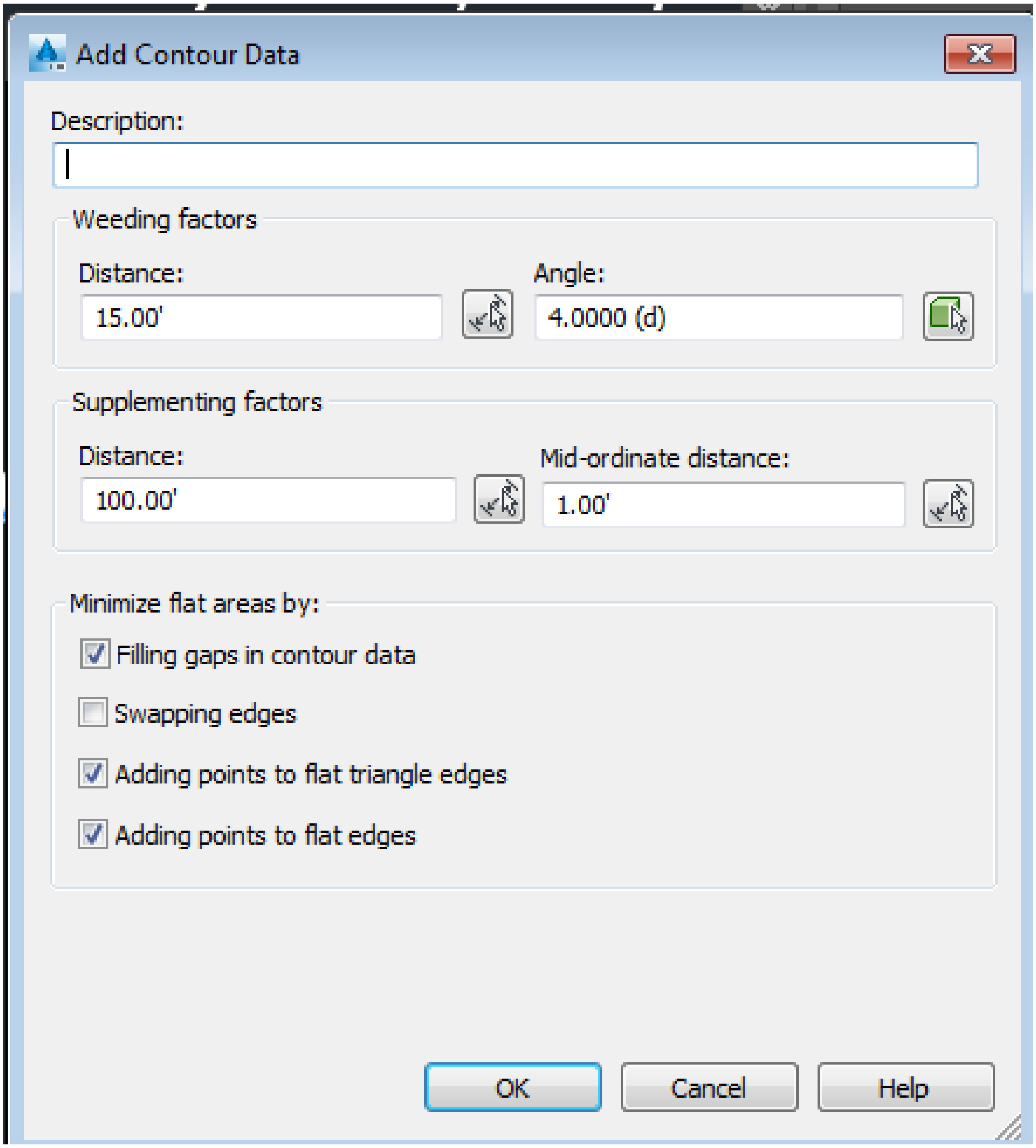

Figure C 11 - Civil 3D contours prompt for defining a surface.

Your surface has now been generated. An example is shown below.

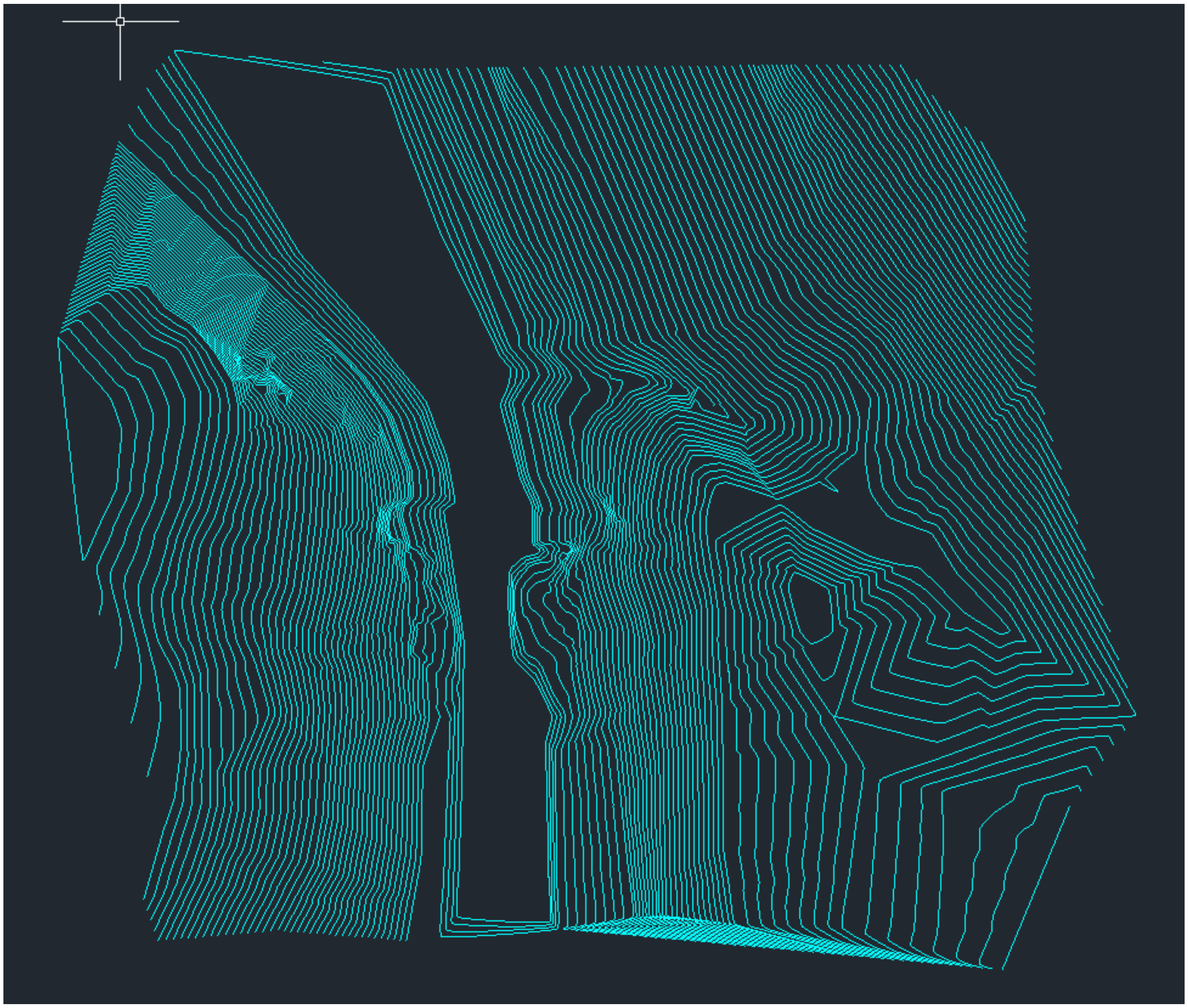

Figure C 12 - AutoCAD Civil 3D surface.

Step 5a – Create Points for Export

Now, draw the boundary you would like to use for the point grid. This can be a rectangle of any size you would like for your model. Make sure it is rectangular. Ensure that the entire surface exceeds the square.

Now, select the Points drop down at the top of the screen, then the Points Creation Tools option:

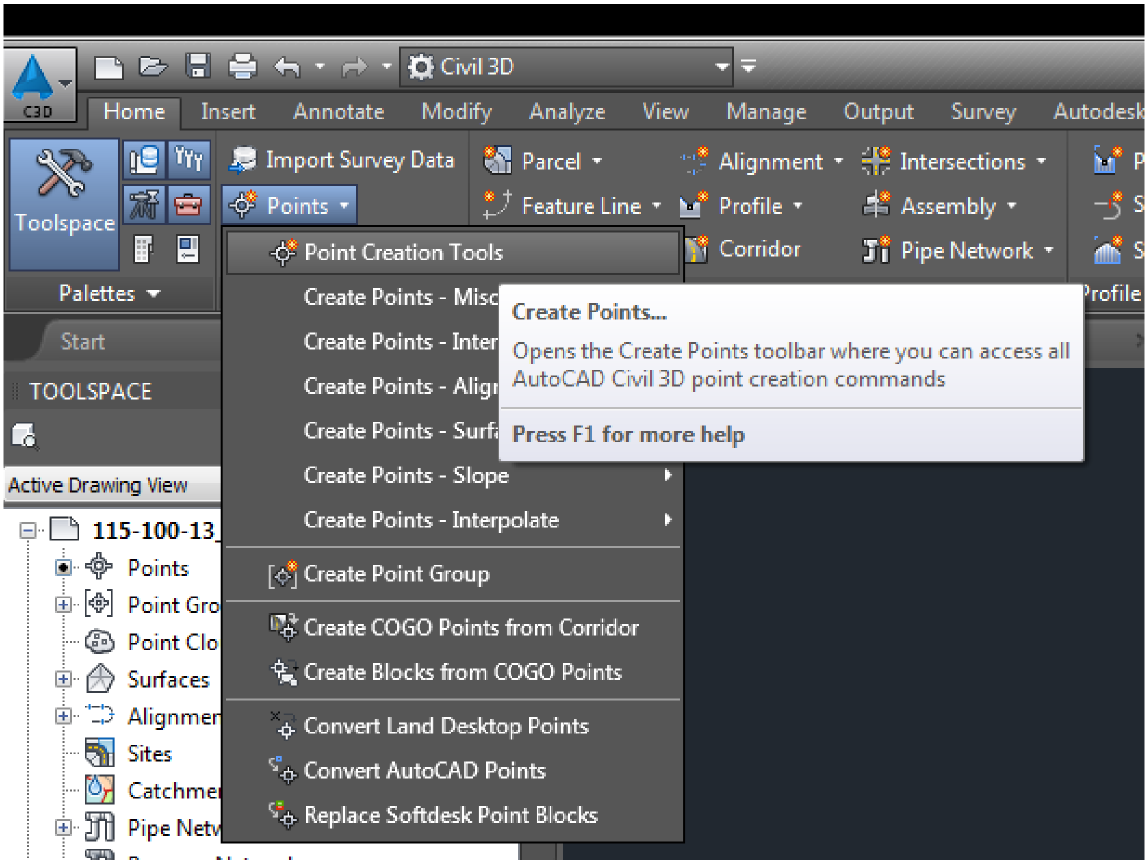

Figure C 13 - Generating points in Civil 3D.

This will open the following dialog box. First, select the down arrow on the right to open the options, then set the options show to "Automatic" or "None" as shown.

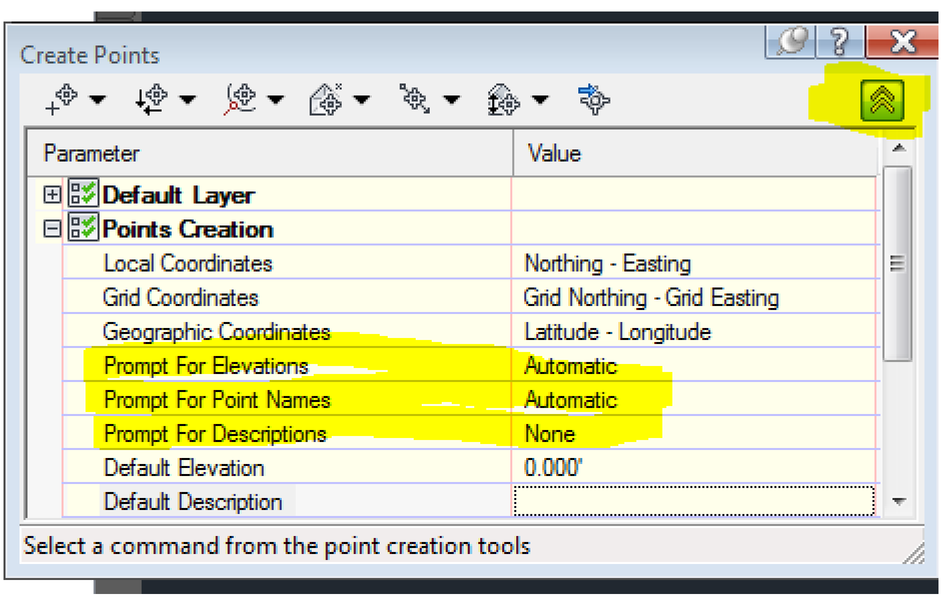

Figure C 14 - Civil 3D point creation definition.

Then, select the drop down arrow on the fourth option from left and select “On Grid”

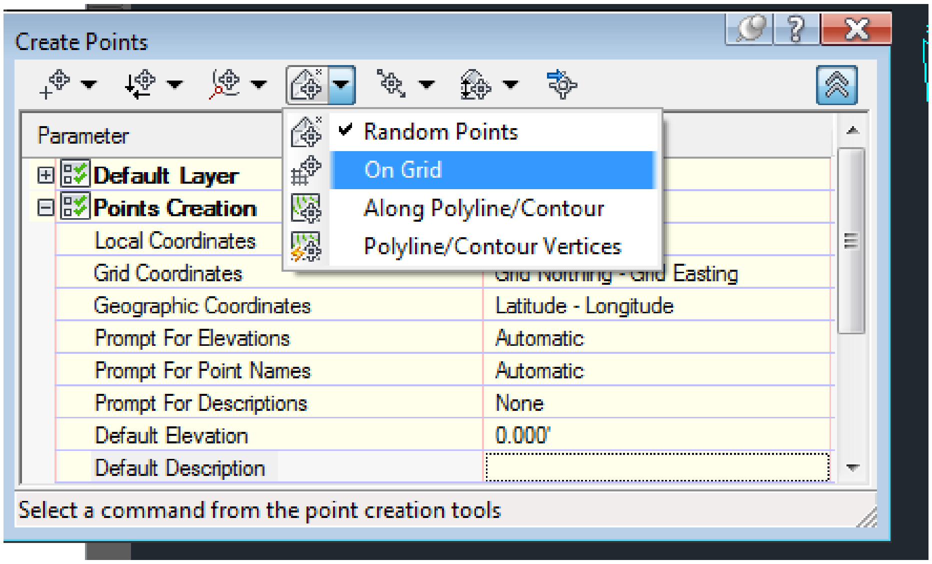

Figure C 15 - Civil 3D point creation "On Grid" selection.

This will start the point command. Do the following:

1. Select the lower left corner of the box
2. Hit enter to choose the default 0 rotation
3. Enter in the desired spacing (ex. 25 ft) and hit enter for X direction spacing
4. Hit enter again to confirm the same spacing for the Y direction
5. Select the upper right corner of the box
6. Hit enter to confirm you DO NOT want to change anything

Done.

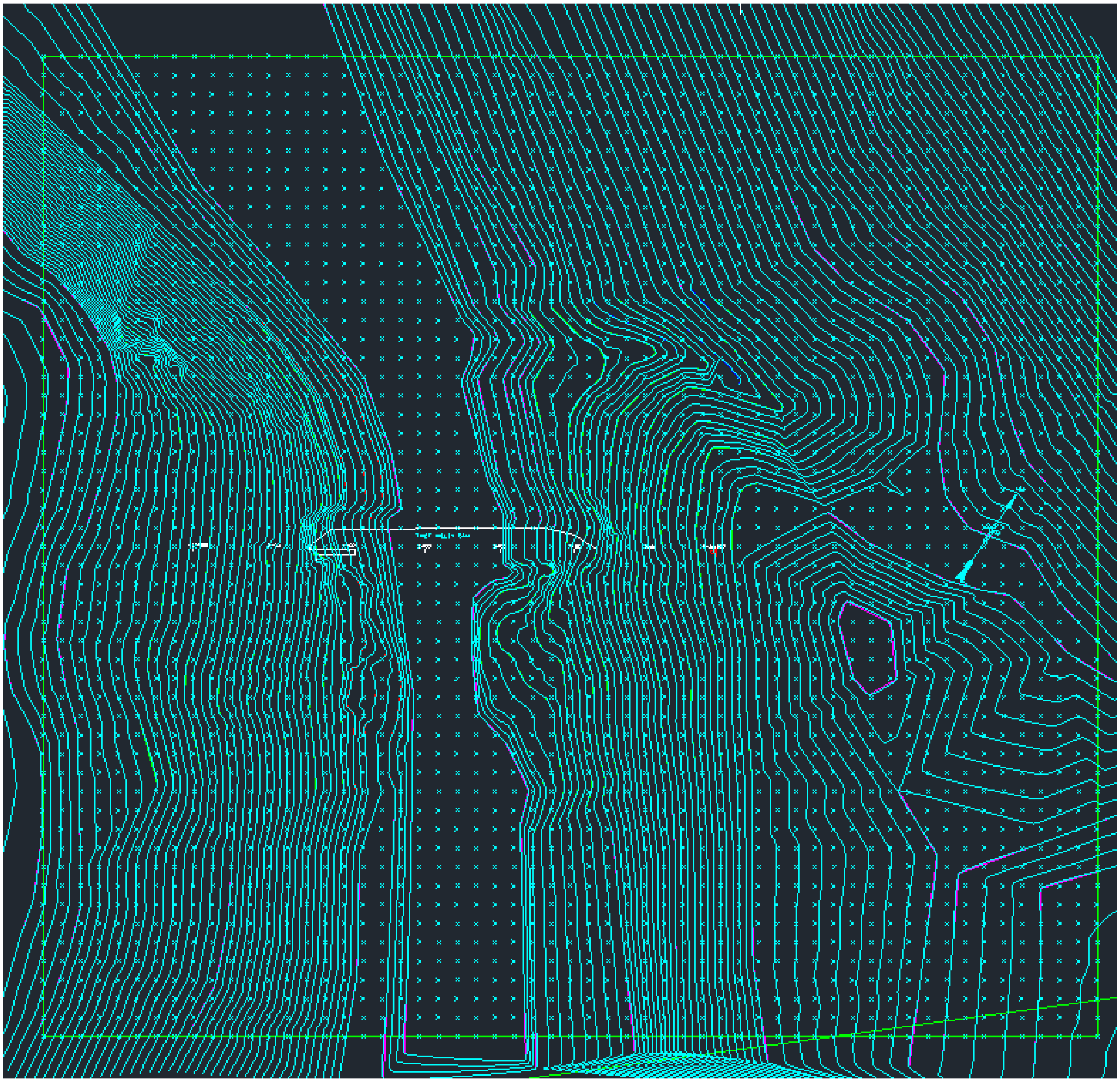

Figure C 16 - Points created on Civil 3D surface using the "On Grid" point creation option.

Step 5b – Generate Contours for Export

Generating contours in the form of polylines to export into an .iges file is simple, but necessary because a Civil 3D surface cannot be exported as an .iges file. Clicking on the desired surface brings up the green surface menu on the ribbon. From that, select the "Extract from Surface" option, then select "Extract Objects."

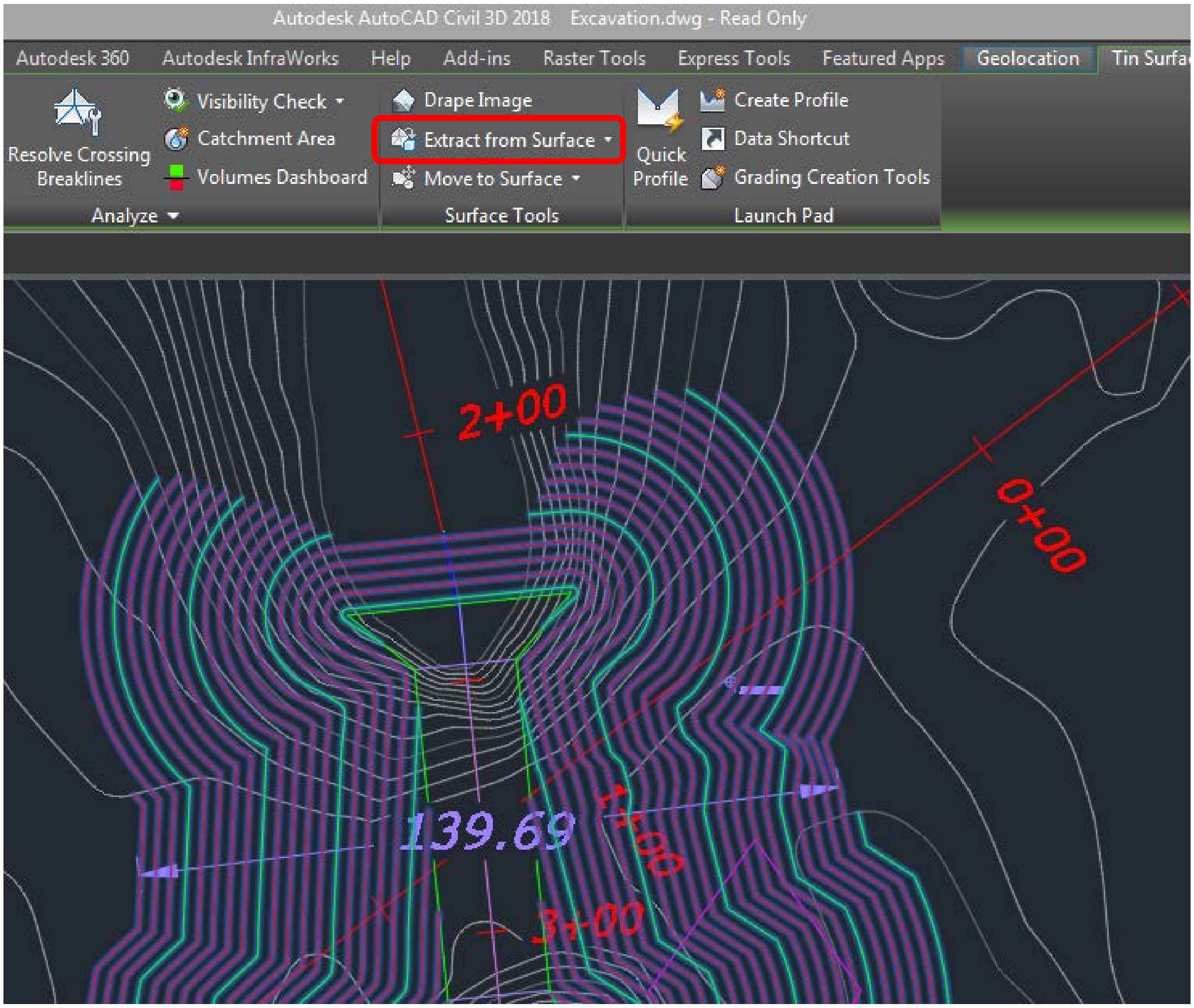

Figure C 17 - "Extract from Surface" option from the surface ribbon.

Following the "Extract Objects" option, the "Extract Objects from Surface" tool box will popup. If all of the contours are desired, make sure "Major Contour" and "Minor Contour" are checked and the drop-down lists say "Select All." Then select "Ok." Otherwise, if only a few contours are desired, select "Select from Drawing" in the drop down, then select the green button to the right of the drop down and select the contours. Close out by selecting "Ok." Polylines of the contours will be located above the surface contour lines. The figure below shows the "Extract Objects from Surface" tool box.

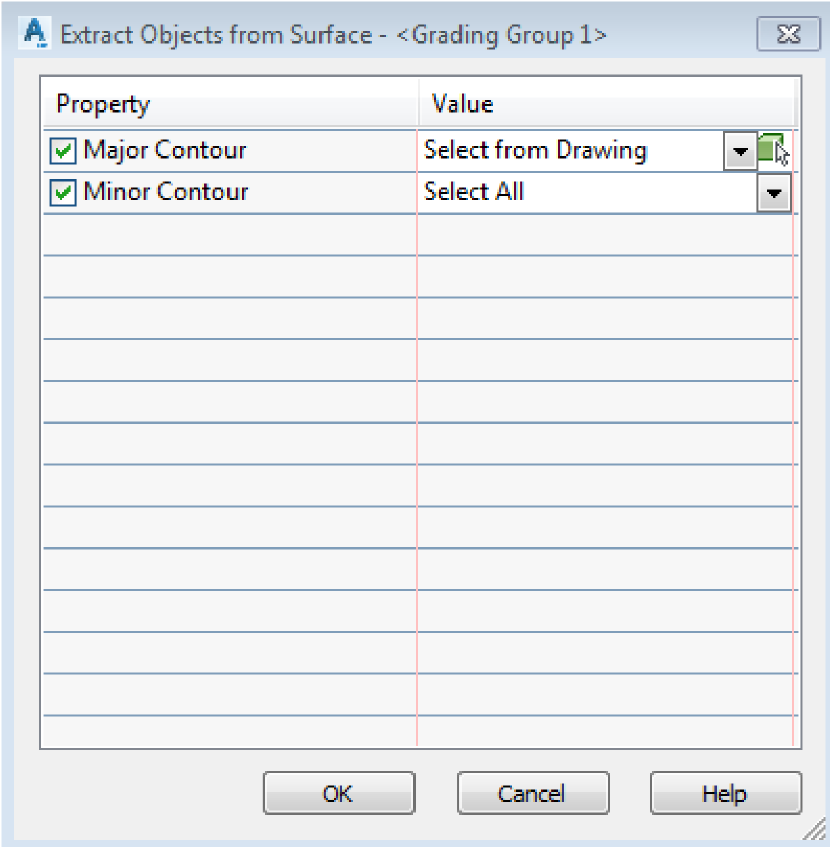

Figure C 18 - Extract Object prompt menu.

Step 6 – Export the Points

Assuming there are no other points in the drawings, we will export all of the points in the file.

In the Toolspace Prospector Tab, right click on Points and select "Export…"

In the new prompt, select "ENZ (space delimited)" and specify the file these will be exported to. Also, uncheck "Do elevation adjustment if possible".

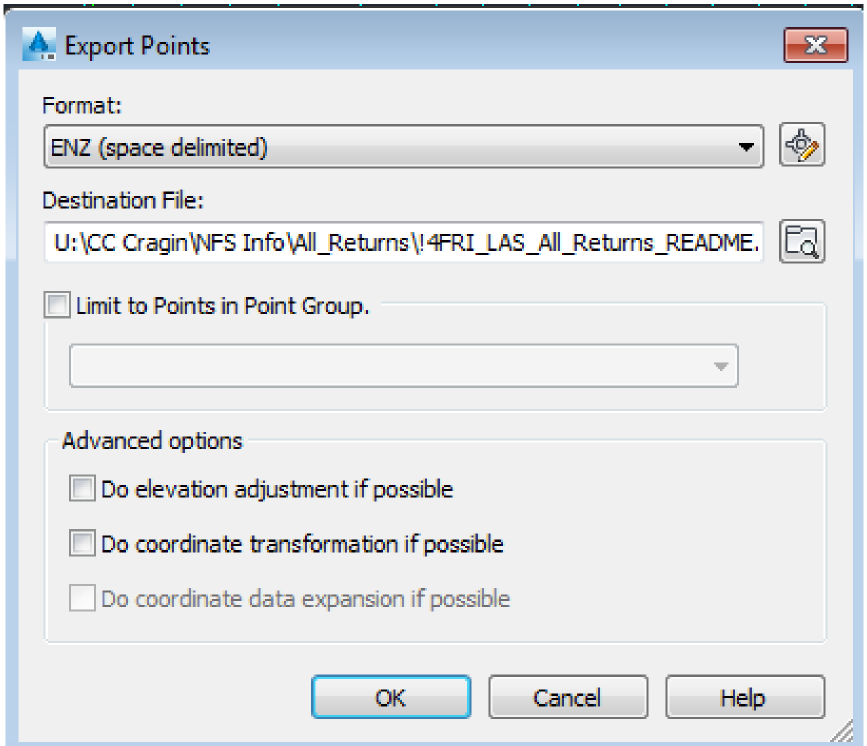

Figure C 19 - Export Points prompt menu.

The points are now in the specified file.

Step 7 – Open the file in Excel

The easiest way to do this is, on the “DATA” tab, on the ribbon, select “From Text”. Then, select the file with the points for import. Specify Delimited in Step 1 of the Import Wizard, then select Space Delimited in Step 2 and select “Finish”.

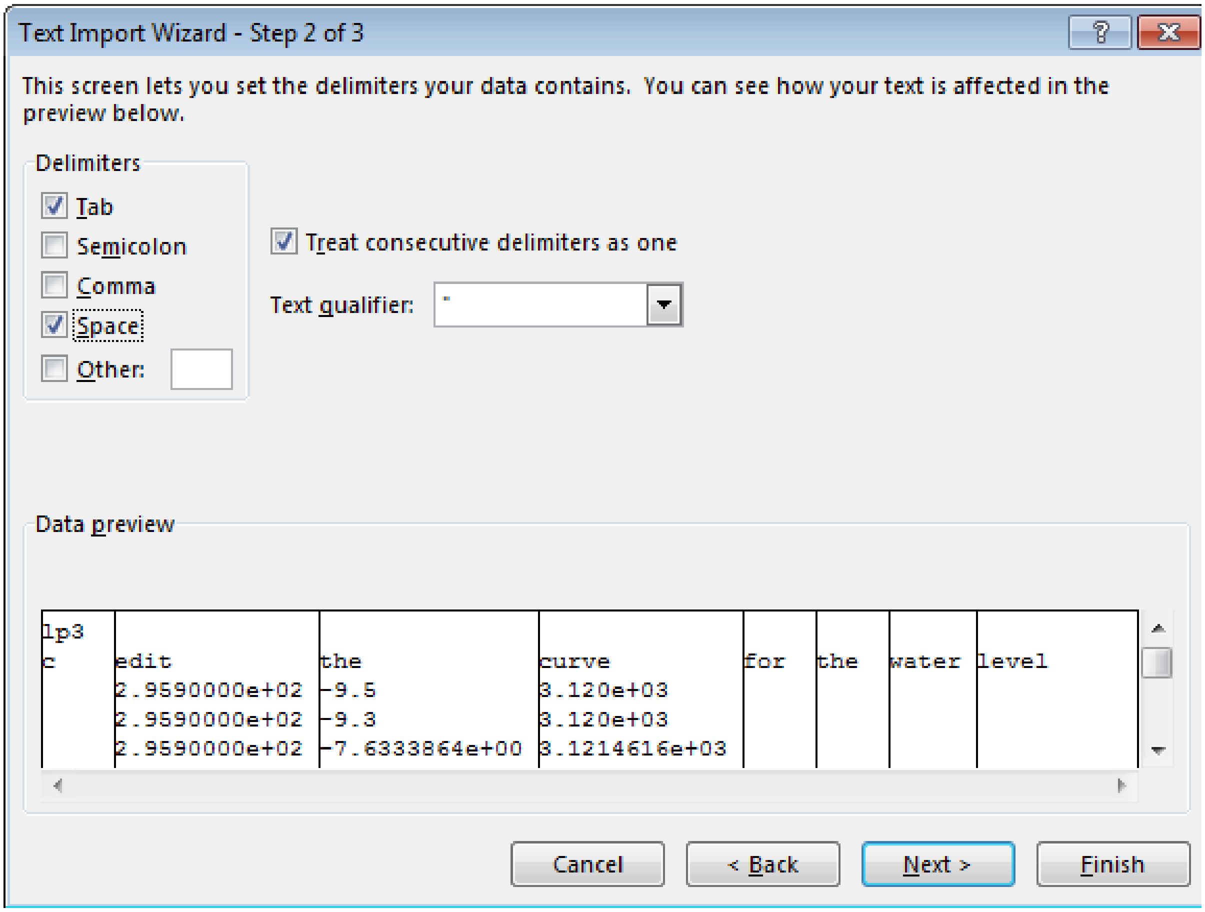

Figure C 20 - Microsoft Excel text import prompt menu.

Step 8 – Format the Data to Import into Code

The surface will be generated using the SD Mesh command in Truegrid. To do this, you need to know the number of rows (unique Northings) and number of columns (unique Eastings) The format of the code input is:

SD (SD#) mesh (unique Northings) (unique Eastings)

(Point 1 Easting) (Point 1 Northing) (Point 1 Elev.)

(Point 2 Easting) (Point 2 Northing) (Point 2 Elev.)

(Point 3 Easting) (Point 3 Northing) (Point 3 Elev.)

…

(Point N Easting) (Point N Northing) (Point N Elev.)

;;

Remember, the coordinates for excel are in global coordinates. Be sure to use a common basepoint with your model to correct the N/E's and bring them toward 0,0. My sheet setup is below:

Easting	Northin	Elevation									CODE				
8904547	501852	3275.7641						Autocad Basepoint			sd	601025	mesh	53	57
8904547	501877	3280.4379				No. of Points		Conversion			-394.555	-644.537	3275.764		
8904547	501902	3283.6634			Northing	53		x	8904941.5554		-394.555	-619.537	3280.438		
8904547	501927	3286.2686			Easting	57		y	502496.5367		-394.555	-594.537	3283.663		
8904547	501952	3288.7184									-394.555	-569.537	3286.269		
8904547	501977	3290.8296									-394.555	-544.537	3288.718		
8904547	502002	3293.2191									-394.555	-519.537	3290.83		
8904547	502027	3295.4142									-394.555	-494.537	3293.219		
8904547	502052	3296.2038									-394.555	-469.537	3295.414		
8904547	502077	3297.3599									-394.555	-444.537	3296.204		
8904547	502102	3298.516									-394.555	-419.537	3297.36		
8904547	502127	3299.1272									-394.555	-394.537	3298.516		
8904547	502152	3299.1272									-394.555	-369.537	3299.127		
8904547	502177	3299.1272									-394.555	-344.537	3299.127		
8904547	502202	3298.9278									-394.555	-319.537	3299.127		
8904547	502227	3298.4082									-394.555	-294.537	3298.928		
8904547	502252	3295.9493									-394.555	-269.537	3298.408		
8904547	502277	3293.278									-394.555	-244.537	3295.949		
8904547	502302	3289.8402									-394.555	-219.537	3293.278		
8904547	502327	3287.5367									-394.555	-194.537	3289.84		
8904547	502352	3286.2915									-394.555	-169.537	3287.537		
8904547	502377	3285.1889									-394.555	-144.537	3286.292		
8904547	502402	3286.6912									-394.555	-119.537	3285.189		
8904547	502427	3287.8258									-394.555	-94.5367	3286.691		
8904547	502452	3288.9604									-394.555	-69.5367	3287.826		

Figure C 21 - Example excel sheet to convert Civil 3D points to Truegrid code.

Now, take the Code portion, save it within your .tg file or in a .geo file and it is ready to use.

REFERENCES

[1] “STL (file format)”, Wikipedia, Wikimedia Foundation, Inc, October, 9, 2017. https://en.wikipedia.org/wiki/STL_(file_format)

[2] “IGES”, Wikipedia, Wikimedia Foundation, Inc, September 7, 2017. https://en.wikipedia.org/wiki/IGES

[3] “Encyclopedia – Definition of: IGES”, PC Mag, Ziff Davis, LLC, 2017. https://www.pcmag.com/encyclopedia/term/44758/iges

[4] *Microsoft Excel*. Vers. 2013. Microsoft, n.d. Web. 2018.