VPG/PrePost Tutorial

Material Thickness Calculation in eta/VPG

Release Date: January, 2007

eta/VPG 3.2

eta

ETA- Engineering Technology Associates, Inc.
1133 E. Maple, Suite 200
Troy, MI 48083

Phone: (248) 729-3010
Fax: (248) 729-3020
Support email: support@eta.com

VPG/PrePost Tutorial

This tutorial was created to familiarize eta/VPG users with the functions and techniques associated with the calculation of part thicknesses, using ETA's M-Step metalforming software within the VPG user environment. This tutorial will demonstrate the steps required to properly prepare a model and assign material thicknesses resulting from the M-Step calculation.

Background Information: Graphic User Interface

The graphic user interface consists of 6 areas within a single window: Main Menu, Drawing Window, View Options, Dialog Window and Display Options and the Top Menu Area.

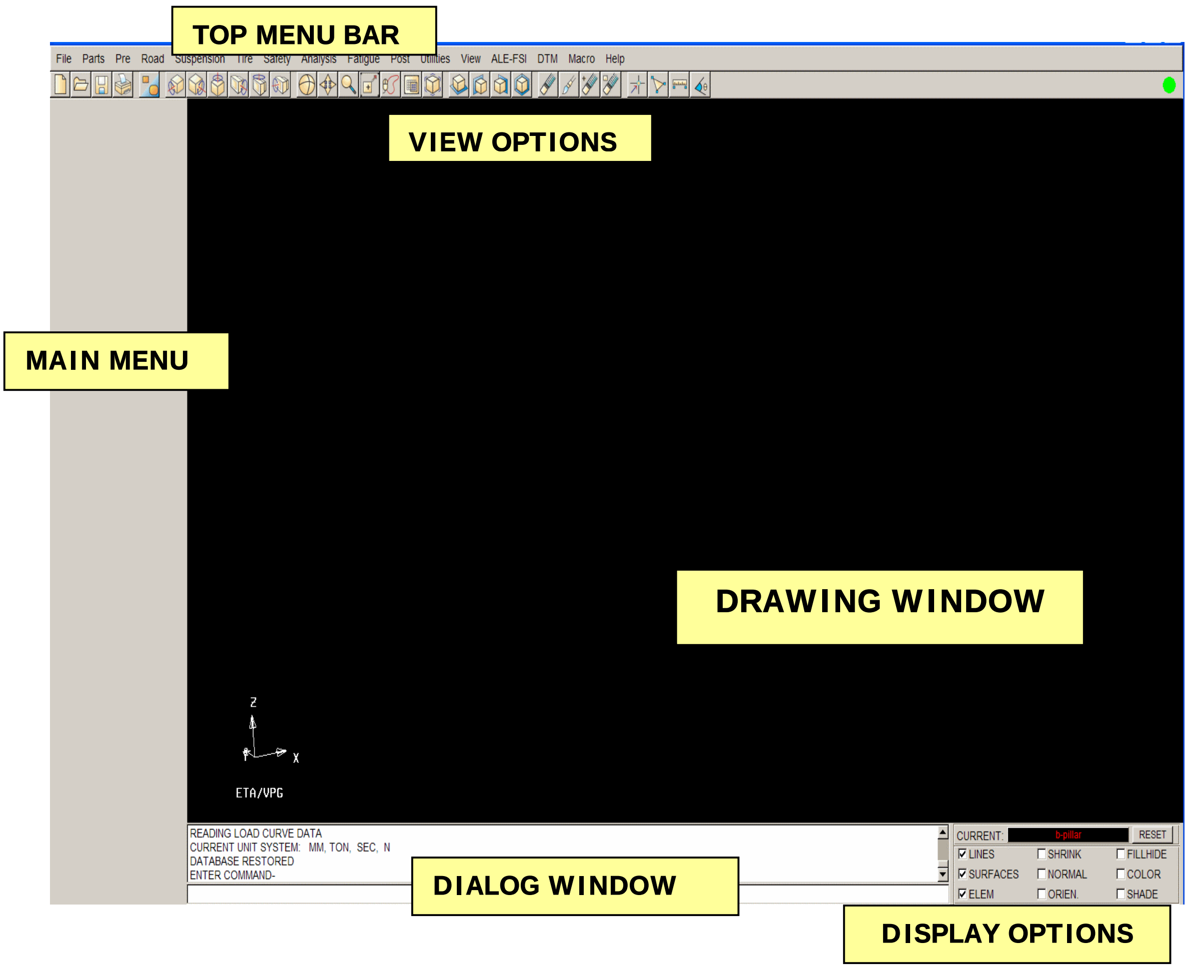


Figure 1: VPG User Interface

1. MAIN MENU: The Main menu options are displayed in this area after selections from the TOP MENU BAR

2. DRAWING WINDOW
   Display graphic entities (CAD- points, lines, surfaces; FEA- nodes, elements).

3. VIEW OPTIONS
   Part control, model positioning, program setup and utilities.

4. DIALOG AREA
   Interface command prompt and scrollable command history.

5. DISPLAY OPTIONS
   Toggle ON/OFF graphic entities to be displayed and controls display mode.

6. TOP MENU BAR
   This menu lists all of the main menus as well as Utility menus and Help Menus

Function Keys

For quick access and to improve productivity, function keys are assigned for the most frequently used menus as follow:

F1	F2	F3	F4	F5	F6
Clear Menu	Element Options	Import File	Line Options	Model Checker	Node Options

F7	F8	F9
Surface Options	Pre-Processor	Online Help

NOTE: Please note that some images shown in this tutorial may make use of optional display features including background colors, shading, and element outlines. These may be controlled through the UTILITY Menu, and the SETUP options found there.

Introduction

Material Thickness variations result from the forming of materials as they undergo plastic deformations, resulting in the desired component shape. This can be especially apparent in thin shell structures using sheet steels or aluminum materials, where it is common to see 10% thinning of the material.

Typical finite element analyses only consider the nominal thickness of sheet components, and do not include any variation in the material thickness or modification of the material property curves due to the plastic deformation.

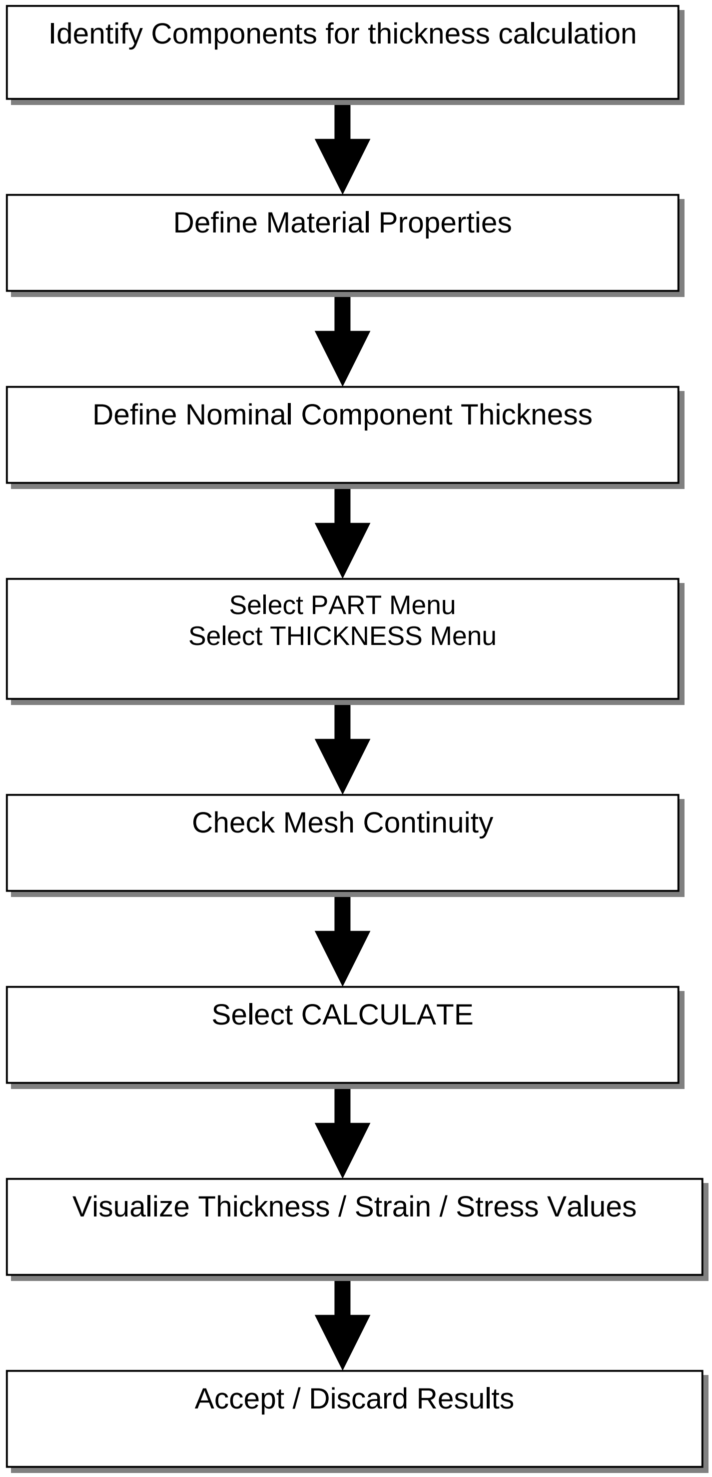
While finite element models achieve higher and higher levels of fidelity due to numerical methods improvements and the ability to include higher levels of design details resulting from computation hardware advances, the influence of manufacturing processes on structural performance has been neglected.

By implementing a simple metalforming simulation tool, VPG can calculate an approximate material thickness and residual plastic strain which would result from forming a flat plate (blank) into a desired shape- the shape of a component.

Including these more localized material properties allows an analyst to increase the accuracy of both durability calculations and nonlinear crash and safety calculations.

This tutorial will guide the user through an example which makes use of the PART THICKNESS function and demonstrates how a model is prepared and demonstrates how the results of the material thickness calculation can be visualized.

Procedural Flowchart



Preparing to use the Thickness Calculation Tool

The user's model should meet several minimum requirements prior to attempting to use the PART THICKNESS command.

- The part(s) must be made up of Shell Elements only
- The part(s) must have defined MATERIAL PROPERTIES, as defined on an LS-DYNA *MATERIAL_XXXX_XXXX entity
- The part(s) must have a defined material thickness, as defined on a *SHELL_SECTION entity.

While the parts must have a consistent element normal, and must a continuous mesh, prior to performing the calculation step, various check/repair features are conveniently available within the PART THICKNESS menu.

Importing a Model File

For this tutorial, we will make use of a model that has already been meshed and has material mechanical properties as well as material thickness defined.

Open a new VPG database by starting VPG and typing a new database name. For our example, please use the name "prt_thickness.vpg".

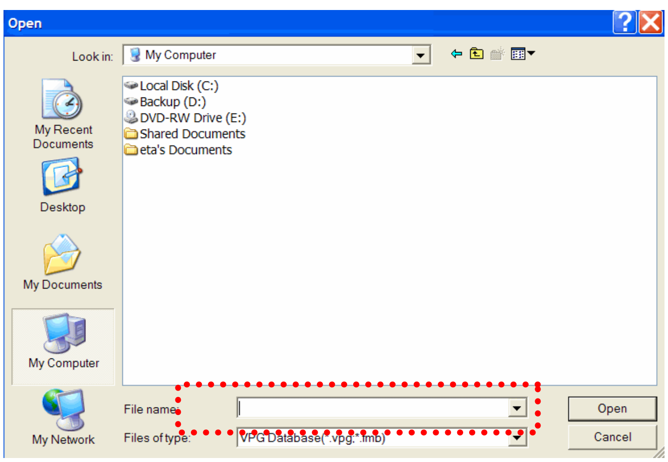


Figure 2: VPG Open File window

VPG will prompt the user to select a solver type. For this example, we will be importing an LS-DYNA file, so select LS-DYNA from the list of solvers presented.

The user must then select a unit system for the model. Please select

1- MM, TON, SEC N.

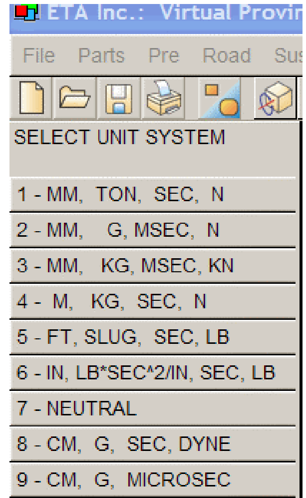


Figure 3: Unit System Selection Menu

Importing an existing LS-DYNA File

To read an LS-DYNA file into VPG:

- Select FILE from the Main Menu (FILE is the menu that users input/output data to or from VPG);
- Select Import or press F3 to read in the tutorial file;
- Select VPG for the LS-DYNA data type (*.dyn) ;
- Select the file **bpillar.dyn**, by either double clicking on it or single clicking and selecting the OK button.

Once loaded, the LS-DYNA model data will be shown as follows (Figure 4):

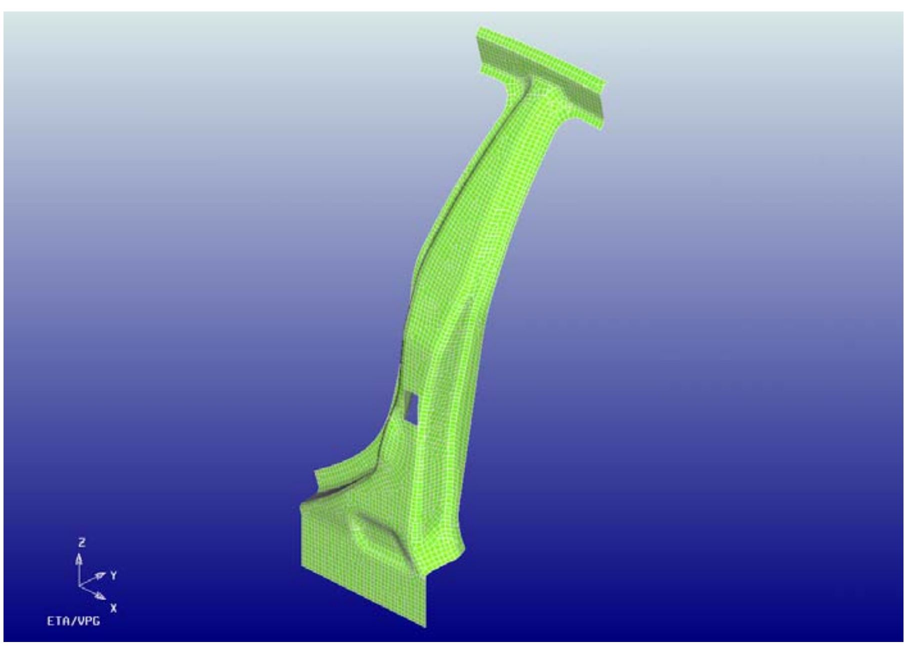


Figure 4: B-Pillar Component Model for Part Thickness Calculation

This model contains a mesh of a part (b-pillar) and has associated material thicknesses, as specified on a *SECTION_SHELL card. This material thickness may be reviewed by entering the ELEMENT PROPERTIES menu and selecting he DEFINE option.

The model also has defined material properties. For purposes of our example, we have defined a simple elastic material model using the *MAT_PIECEWISE_LINEAR_PLASTICITY model with common mild steel properties. These material properties can be reviewed by selecting MATERIAL from the PRE Menu.

Using the PART THICKNESS Function

To access the Part Thickness menu, the user selects the PART menu from the top Menu Bar. This action will load the PART menu in the Main Menu Display area.

The PART Menu contains all actions and tools associated with the creation, modification, manipulation and display of parts within the model database. The complete PART menu is shown in Figure 5.

Select THICKNESS to enter the part thickness calculation menu. This menu contains functions associated with 1) checking the model prior to performing the calculation 2) executing the calculation 3) displaying the results of the calculation 4) removing (discarding) the results.

The THICKNESS MENU, shown in Figure 6, has the following functions:

CHECK CONTINUITY: This function checks the mesh to determine if the elements in the part form a continuous mesh. VPG cannot calculate thickness and stress/strain for meshes that are discontinuous. If this mesh does not meet the requirements for the calculation, VPG responds with the message “selected mesh is discontinuous” and stops the operation.

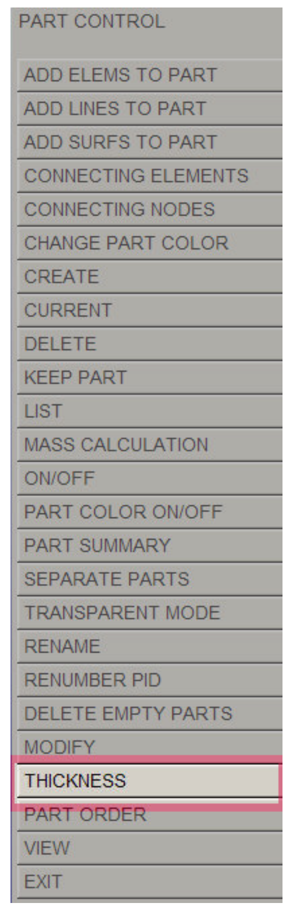


Figure 5: Part Menu

NOTE: Observe messages in the DIALOGUE window for guidance during these operations. See **Creating a PART THICKNESS** (pg. 11) for more details on this and the following executions.

AUTO NORMAL: This function checks for consistency of all element normals in the part(s) selected. Elements contained within the same part must have consistent normal definitions. Elements Normals are defined by the order that the nodes are specified in the element definition. VPG will detect elements whose normals are not consistent with a selected "seed" element and will revise the node order in the element definitions when necessary to make all the normals consistent.

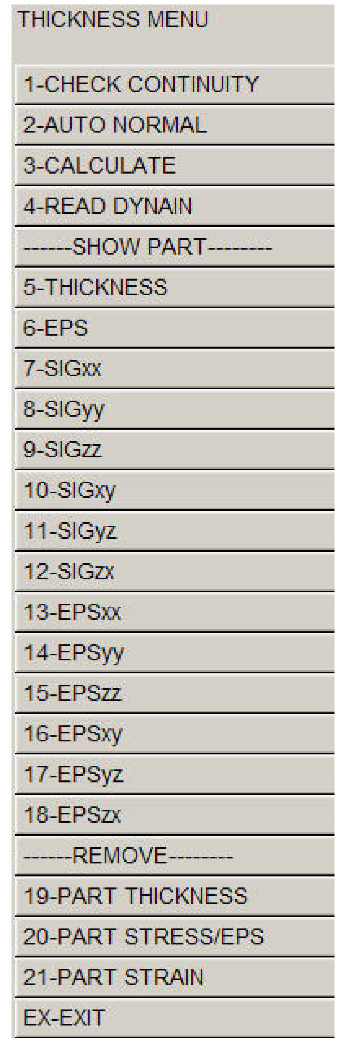


Figure 6: Part THICKNESS Menu

CALCULATE: This function executes the M-STEP program in a separate, temporary window. VPG automatically transfers the data into to the M-STEP program and executes a geometric metal forming simulation.

> Click on CALCULATE then select the part.

At the completion of the calculation, the window will close and return a message:

"CALCULATE THICKNESS OF PICKED PART
UPDATE THICKNESS OF PICKED PART"

in the message /prompt area.

READ DYNAIN: The DYNAIN file is an optional way to attain the material thickness and residual plastic strain data. By selecting this function, the user may specify an existing DYNAIN file to be included in the model definition.

If the mesh definition is not consistent with the mesh of the target part, VPG will interpolate the node thickness values and apply the value to the nearest node. This feature is important if an adaptive mesh was used in the metal forming analysis, which produced the DYNAIN file.

SHOW PART-THICKNESS: Selection of this function generates a contour plot showing the material thickness as colors, indexed to a legend appearing on the right side of the drawing window.

> Select the part then select Exit to display the contour.

Selecting the DISPLAY OPTION function may modify the results displayed. Using this menu, you may change the range, number of contour levels, shading options, and scale values.

SHOW PART – EPS: This function generates a contour plot showing the total plastic strain resulting from the metal forming operation. In a similar manner the EPSxx, EPSyy, EPSzz, EPSxy, EPSyz and EPSzx all display contour plots of the corresponding strain component.

SHOW PART – SIGxx, SIGyy, SIGzz, SIGxy, SIGyz, SIGzx: These functions generate contour plots showing the corresponding stress component.

REMOVE PART THICKNESS: This function restores the original uniform part thickness and discards the calculated variable part thickness.

REMOVE PART STRESS/EPS: This function removes the calculated stress and plastic strain for each element and restores the original value of zero.

REMOVE PART STRAIN: This function removes the calculated strain for each element and restores the original value of zero.

EXIT: Closes the PART THICKNESS menu and returns to the PART menu.

Creating a PART THICKNESS

On the THICKNESS menu, select CHECK CONTINUITY. A part list will be opened, and VPG will request the user to select the part to be checked. Select the part “b-pillar” from the list or by selecting an element in the part in the display window. The selected part will be highlighted. When the parts have been selected, the user then completes the selection list by clicking DONE.

VPG will check the mesh to assure that the mesh is continuous and there are no unconnected elements in the selected part(s).

If the mesh is continuous, VPG will respond, “**part passes the check**”, and the user may then proceed to other operations. If the part(s) fail the check the message “part fails check / does not continue”, and the operation will terminate. The user should then review the model using MODEL CHECK / BOUNDARY to assure that the mesh is suitable for use in this function.

HINT:
If the part is made up of two complete components that happen to be grouped in a single part, the user should place these in separate parts using a element move/copy commands. An example of this condition might be two symmetric parts which are identical, but are mirrored about a plane of symmetry. For these parts to make use of the thickness calculation tool, they should be placed in separate parts

On the THICKNESS menu, select AUTO NORMAL. VPG will prompt the user to select an element for normal checks.

After the part is selected, an arrow is displayed indicating the direction of the element normal as determined by the order of the node definitions (using a right hand rule). If the user would like to accept the normal direction he may indicate this by selecting **YES**. If the user would like to reverse the normal direction he may indicate this by selecting **NO** and the element normal(s) will be reversed. At this time, all elements in the part will be compared to the selected element, and the opposing element normals will be reversed automatically. These elements will be highlighted and a message will be displayed “XXX element normals reversed” in the message/prompt area.

For our example, all element normals should already be consistent and a message “all normals consistent” will be displayed in the message/prompt area.

To execute the calculation of thickness and stress/strain, select the CALCULATE function from the menu. When this is selected, a DOS window will open and displayed messages associated with the M-STEP solver used to make these calculations.

After completion of the M-STEP calculation, the window will automatically close. Any error messages will be displayed in the message/prompt area. After successful completion the message, “ calculation completed” will be displayed. Otherwise, a “calculation failed” message will be displayed.

Viewing the Results

Select THICKNESS from the SHOW PART portion of the THICKNESS menu.

VPG will display a part list and VPG will request the user to select the part(s) to be checked. Select the part its name (in this case, ***b-pillar*)** from the list or by selecting an element in the part in the display window. The selected part will be highlighted. When the parts have been selected the user then completes the selection list by clicking EXIT.

A contour plot of the material thickness, as calculated by VPG will be displayed. Figure 7 shows the contour image of the b-pillar of our example.

The contours shown in the image correspond to the legend on the right side of the display window. Selecting the DISPLAY OPTIONS menu item allows the user to change the display of the contours. This will open a window where the contour range, max/min values, number of contours, element outline and shading options can be controlled.

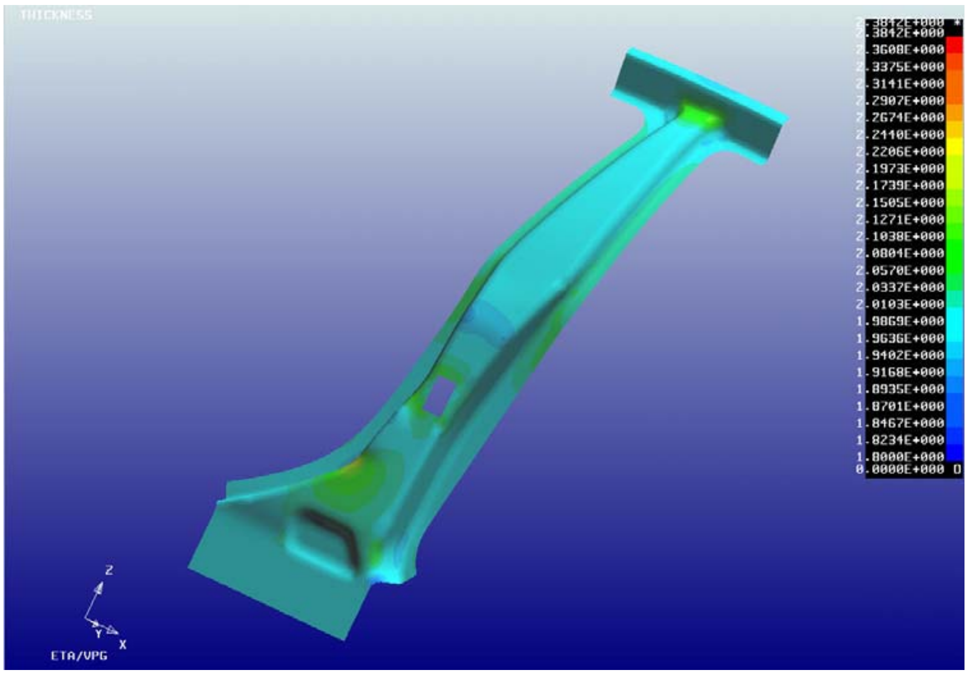


Figure 7: Thickness Contour Plot of the example problem

Select EPS from the SHOW PART menu. The image shown in Figure 8 will be displayed. This contour shows the Plastic Strain, which has been calculated, based on the material properties supplied and the rotation of each element with respect to the adjacent elements and the flat blank shape.

The figure may be manipulated using the rotation/zoom/ pan icons (Top Menu Bar).

If these results are acceptable to the user, they may EXIT the Part THICKNESS menu and the results will be added to the model. If these results are not acceptable to the user, they may discard a portion or all of the results using the REMOVE PART THICKNESS/STRESS / STRAIN functions, where all calculated values will be discarded from the model.

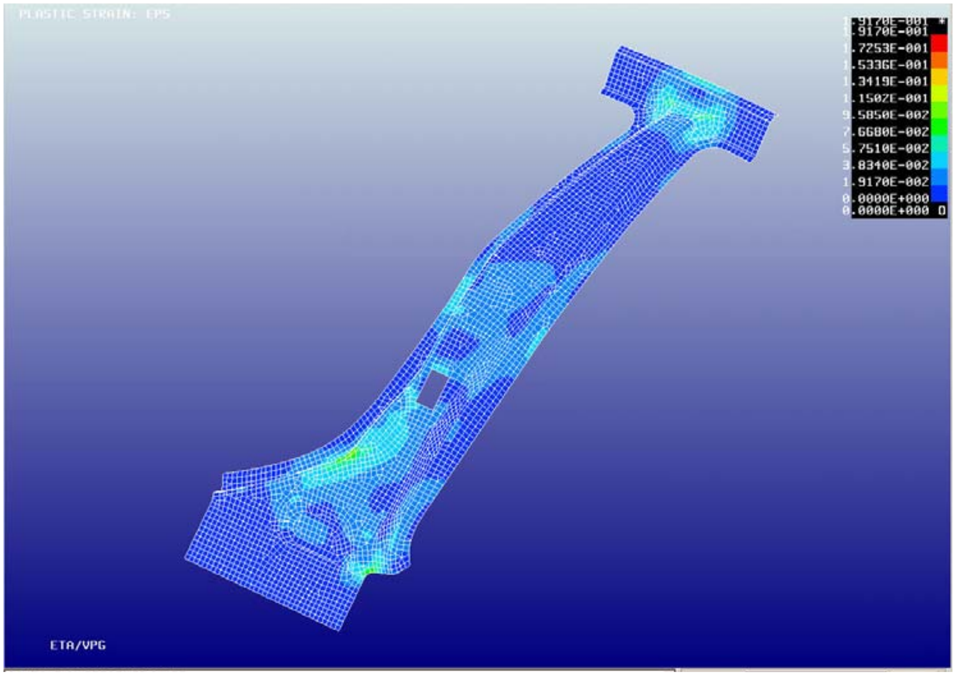


Figure 8: EPS Contour Plot from Part Thickness Calculation

The resulting model, when exported in DYNA format will contain the initial strain data and the material thickness data in the form shown in Figure 9. All strain values will be implemented through the *INITIAL_STRAN_SHELL.

```
       0.0       0.0       0.0       0.0       0.0       0.0       0.0       0.0
$      es1       es2       es3       es4       es5       es6       es7       es8
       0.0       0.0       0.0       0.0       0.0       0.0       0.0       0.0
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
*INITIAL_STRAIN_SHELL
$      eid
         1
$    epsxx     epsyy     epszz     epsxy     epsyz     epszx
   0.01101   0.00341  .3132E-3  -0.04496   0.00133  -0.00659
$    epsxx     epsyy     epszz     epsxy     epsyz     epszx
   0.01101   0.00341  .3132E-3  -0.04496   0.00133  -0.00659
*INITIAL_STRAIN_SHELL
$      eid
         2
$    epsxx     epsyy     epszz     epsxy     epsyz     epszx
  .2088E-3 -.4980E-4 -.2970E-5  .3134E-3 -.1277E-4 .54560E-4
$    epsxx     epsyy     epszz     epsxy     epsyz     epszx
  .2088E-3 -.4980E-4 -.2970E-5  .3134E-3 -.1277E-4 .54560E-4
*INITIAL_STRAIN_SHELL
$      eid
         3
$    epsxx     epsyy     epszz     epsxy     epsyz     epszx
   0.00115  -0.00254 -.8960E-4 .47550E-4 -.4770E-3 .44980E-6
$    epsxx     epsyy     epszz     epsxy     epsyz     epszx
   0.00115  -0.00254 -.8960E-4 .47550E-4 -.4770E-3 .44980E-6
*INITIAL_STRAIN_SHELL
$      eid
         4
```

Figure 9: Initial Plastic Strain from VPG Thickness Calculation

Element thicknesses will be implemented through the *ELEMENT_SHELL_THICKNESS entity. The format of these thicknesses are shown in Figure 10.

```
    4755 3.584302490E+03-5.610137939E+02 1.587777588E+03
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
*ELEMENT_SHELL_THICKNESS
$    eid     pid      n1      n2      n3      n4
       1       2       1       2     274      62
$           thic1           thic2           thic3           thic4             psi
 1.970756800E+00 1.970756800E+00 1.970756800E+00 1.970756800E+00 0.000000000E+00
       2       2       6       7       8    2023
 1.999688000E+00 1.999688000E+00 1.999688000E+00 1.999688000E+00 0.000000000E+00
       3       2       5       6    2023    2024
 2.002950700E+00 2.002950700E+00 2.002950700E+00 2.002950700E+00 0.000000000E+00
       4       2       4       5    2024    2025
 2.000793000E+00 2.000793000E+00 2.000793000E+00 2.000793000E+00 0.000000000E+00
       5       2    4746       4    2025      64
 2.004850600E+00 2.004850600E+00 2.004850600E+00 2.004850600E+00 0.000000000E+00
       6       2       8       9    2026    2023
 2.000519800E+00 2.000519800E+00 2.000519800E+00 2.000519800E+00 0.000000000E+00
       7       2       9      10    2027    2026
 2.003803500E+00 2.003803500E+00 2.003803500E+00 2.003803500E+00 0.000000000E+00
       8       2      10      11    2028    2027
 2.004499700E+00 2.004499700E+00 2.004499700E+00 2.004499700E+00 0.000000000E+00
       9       2      11      12    2029    2028
 2.002688900E+00 2.002688900E+00 2.002688900E+00 2.002688900E+00 0.000000000E+00
      10       2      12      13    2030    2029
 2.004787400E+00 2.004787400E+00 2.004787400E+00 2.004787400E+00 0.000000000E+00
      11       2      13      14    2031    2030
 2.002819300E+00 2.002819300E+00 2.002819300E+00 2.002819300E+00 0.000000000E+00
      12       2      14      15    2032    2031
```

Figure 10: Element Shell Thickness from VPG Thickness Calculations

Conclusion

Eta/VPG can automatically calculate and assign localized material thickness due to manufacturing processes such as stamping. These results can be included in LS-DYNA files through the *ELEMENT_SHELL_THICKNESS and *INITIAL_STRAN_SHELL.

References

Users may find additional information on the M-Step solver and a theoretical overview of the thickness and stress / strain calculation by reviewing the M-Step Introduction Manual. The manual is available from the ETA ftp site.