

eta/DYNAFORM

BSE Training Manual

Version 5.5



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

Tel: (248) 729-3010

Fax: (248) 729-3020

Email: support@eta.com

eta/DYNAFORM team

September 2006

FOREWORD

The concepts, methods, and examples presented in this text are for illustrative and educational purposes only, and are not intended to be exhaustive or to apply to any particular engineering problem or design.

This material is a compilation of data and figures from many sources.

Engineering Technology Associates, Inc. assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

TABLE OF CONTENTS

I. INTRODUCTION TO BSE	1
II. Create DataBase and Read in Model File	2
III. Edit Parts in Database	4
IV. Mesh Generation.....	5
V. Mesh Check.....	9
VI. MSTEP Module and Parameter Setup	12
VII.Start up Post-Processor and Analyze Simulation Result	18
VIII.Blank Nesting.....	22

I. INTRODUCTION TO BSE

BLANK SIZE ENGINEERING (BSE) is an eta/DYNAFORM add-on module. The functions provided in the BSE submenu are designed to unfold a part and estimate a flat blank outline. In addition, BSE can be utilized to estimate a blank size, conduct a blank development and calculate material utilization. As shown in Figure 1.1, the BSE menu consists of PREPARATION, MSTEP, and DEVELOPMENT.



Figure 1.1: BSE menu

The *MSTEP* is a modified one step solver that allows users to perform both blank size estimate and quick formability analysis. Figure 1.2 illustrates the streamlined process guidance graphic user interface (GUI) of *MSTEP*.

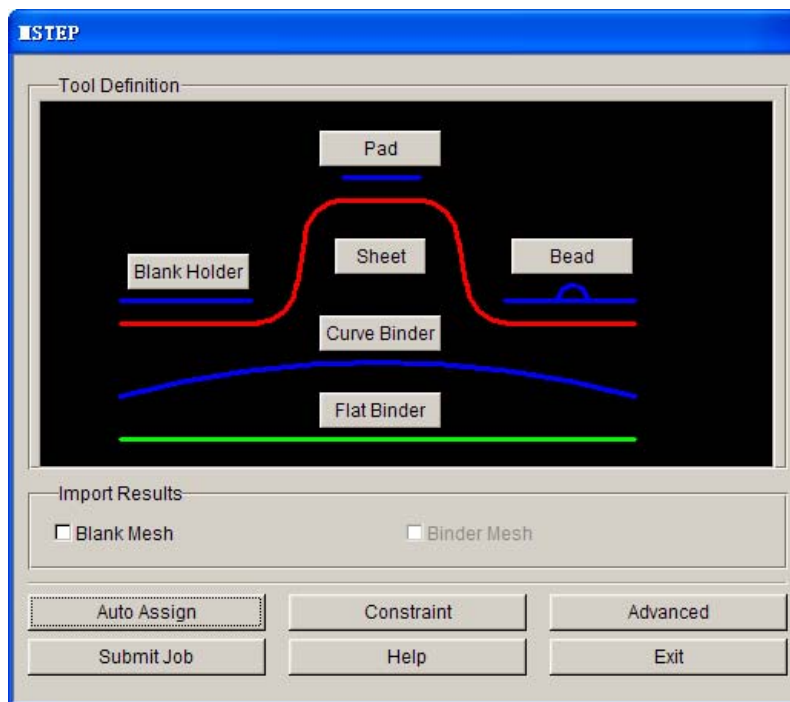


Figure 1.2: MSTEP GUI

The “Advance” function in MSTEP allows users to select either Fast or Accurate option. The *Fast* option enables rapid calculation to obtain blank outline from a given part geometry, which the *Accurate* option provides both blank outline and quick formability analysis.

After obtaining the results, the *Postprocessor* is used to view the thickness, thinning, stress and strain of the sheet metal part. Option for displaying FLD is also available.

The *Development* menu enables the users to perform blank development and blank nesting operation using the blank outline obtained from *MSTEP*.

Advantage of utilizing BSE module in part feasibility study:

- Enables Tool Makers to develop the Blank, estimate the Blank Size and costs for material
- Enables Tool Makers to generate the nest and estimate the tooling costs per strip layout
- Eliminates the time consuming, manual process of sectioning and flattening a sheet metal part

II. CREATE DATABASE AND READ IN MODEL FILE

Start eta/DYNAFORM 5.5

For workshop and Linux user, enter “df55” (default) order to start eta/DYNAFORM 5.5. For PC user, double click eta/DYNAFORM 5.5 (DF55) icon or select DYNAFORM from program startup to start the software.

After starting eta/DYNAFORM, program automatically creates an empty database named **Untitled.df**. You continue by importing CAD or CAE model to the database to begin the practice.

Import File

1. Click **BSE→Preparation** from the **Menu bar** to display the **BSE Preparation** dialog box shown in Figure 2.1. Select the **IMPORT** function. The **Import file** window illustrated in Figure 2.2 is displayed.

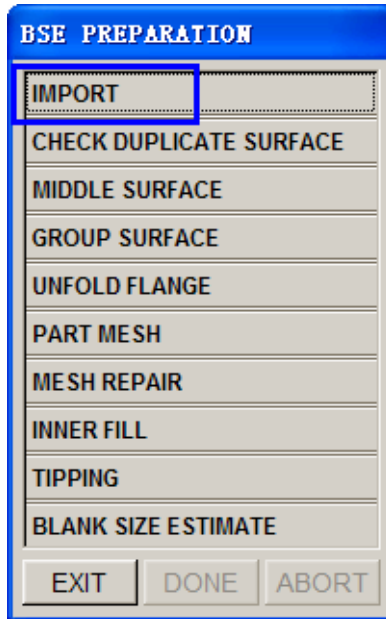


Figure 2.1: BSE preparation dialog box



Figure 2.2: Import file window

Locate the CAD data example1.igs from the directory. Then, use your mouse cursor to pick the file. Next, click OK button to import the data into eta/DYNAFORM database.

After importing the file, check if the displayed model resembles Figure 2.3. The model is displayed in the screen display area in isometric view. This view is the default setting of eta/DYNAFORM.

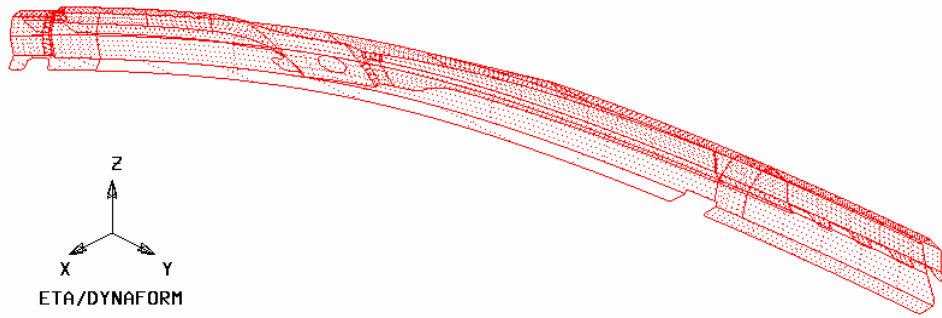



Figure 2.3: Illustration of Example 1

Note: Icons are different in different system platform. Functions of other icons are introduced in the following chapters. You may also refer to eta/DYNAFORM User's Manual for description of all functions.

2. Save database file to the assigned working directory. Select menu **File**→**Save as** or  icon from **Icon bar**. After inputting “example1.df”, select **Save** button to save the database and exit the window.

Refer to eta/DYNAFORM User's Manual for detailed information of eta/DYNAFORM database unit system and file type.

III. EDIT PARTS IN DATABASE

In eta/DYNAFORM, all models are managed based on parts. Under default condition, every entity is created or read into part. Refer to eta/ DYNAFORM User's Manual for detailed information about part manager.

As shown in Figure 2.4, the **Edit** function in **Parts** manager is used to edit part properties and delete parts.

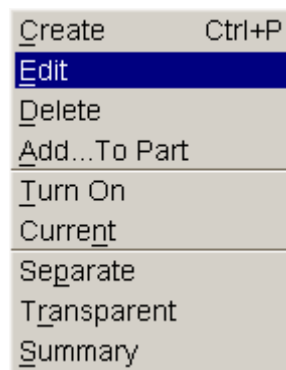


Figure 2.4: Part menu

1. Select **Parts**→**Edit** to display the **Edit Part** dialog box. All defined parts are displayed in the list. Parts are marked with name and identification number. User may change part name and identification number and at the same time may delete parts from database.
2. Select **C001V000** from part list shown in Figure 2.5. In the Name input field, enter **BLANK** following by clicking **Modify** button in the lower left corner of the dialog box to complete the operation. You may also change the part color.
3. Click **OK** button to dismiss the dialog box.

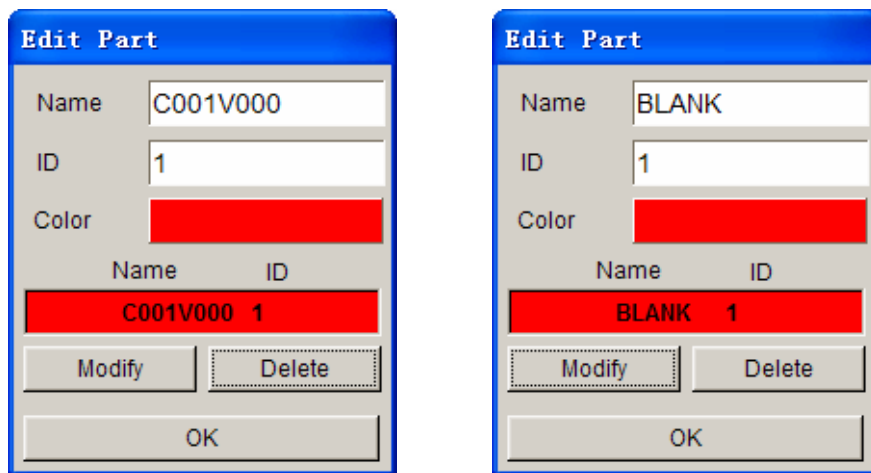


Figure 2.5: Edit part dialog box

IV. MESH GENERATION

Most meshes in eta/DYNAFORM are generated using **Surface Mesh** function. It is a rapid and robust meshing tool which automatically creates mesh based on CAD surface data. For detail functions about **Surface Mesh** function, refer to eta/DYNAFORM User's Manual.

1. Select **BSE**→**Preparation**→**PART MESH** function. See Figure 2.6.

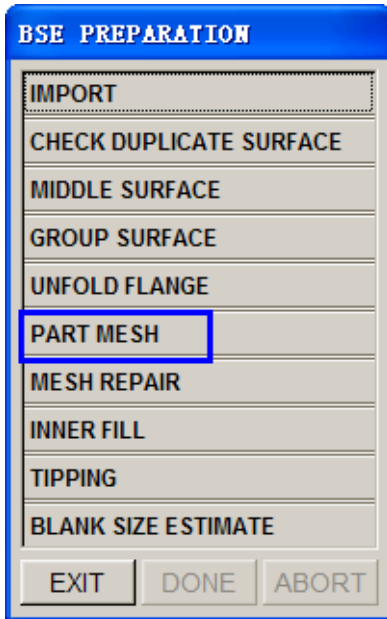
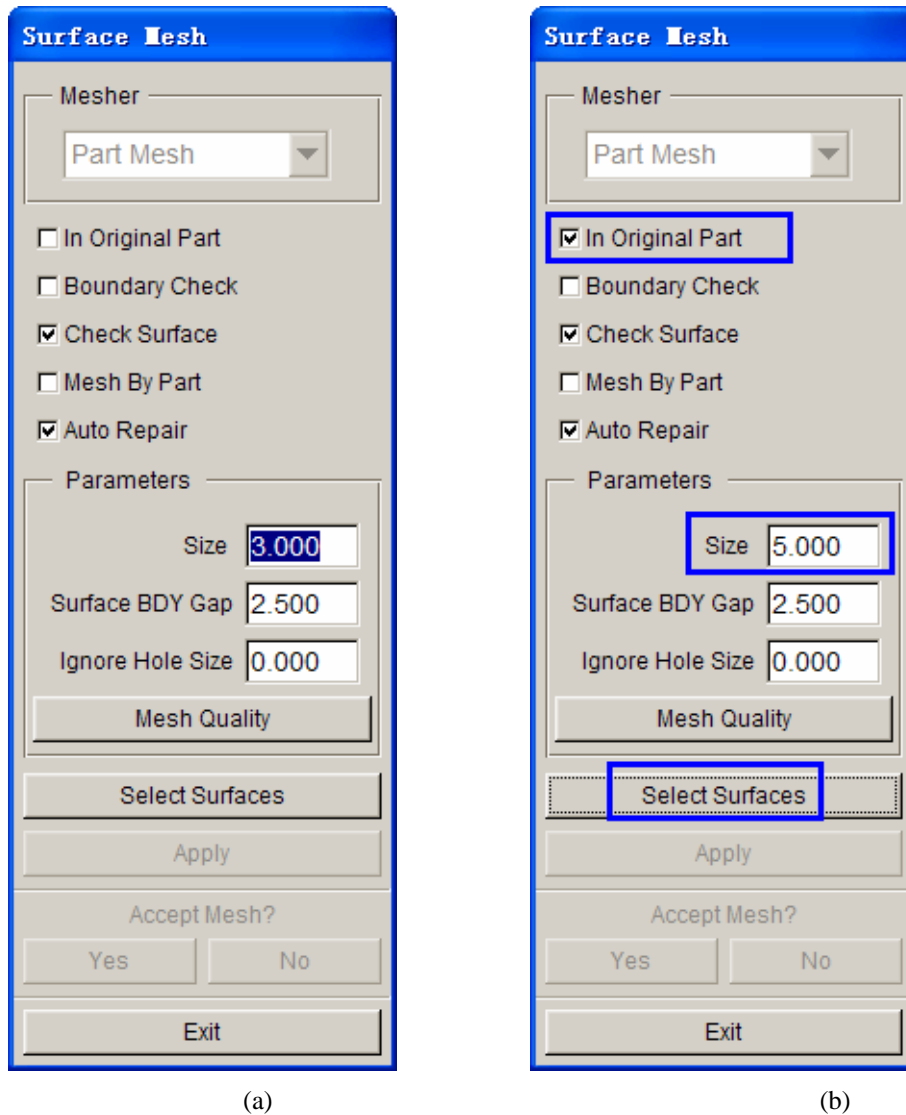


Figure 2.6: BSE preparation dialog box

2. **Surface Mesh** dialog box shown in Figure 2.7a is displayed after you select the **PART MESH** function.



(a)

(b)

Figure 2.7: Surface mesh dialog box

3. From the dialog box, toggle on checkbox of **Original Part** option. Next, change the mesh size to **5.000** (mm), while keeping other options in default setting. See Figure 2.7b.
4. Select **Select Surfaces** button from **Surface Mesh** dialog box.
5. Select **Displayed Surf** Button in **Select Surface** dialog box illustrated in Figure 2.8.
6. Pay attention to all current displayed surfaces which are highlighted in white. This indicates they are all selected. The **Select Surface** dialog box provides different methods for selecting surfaces, place your mouse cursor on each button to view name of each icon.

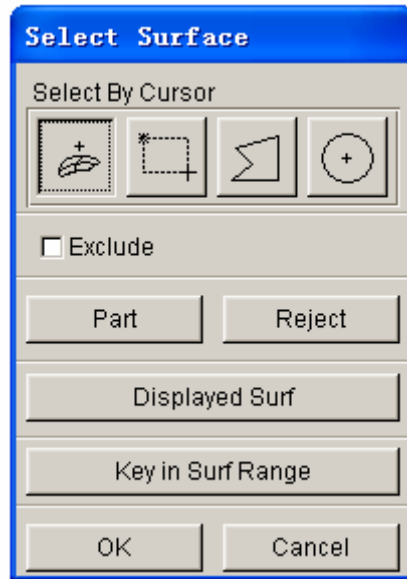


Figure 2.8: Surface mesh dialog box

7. Select **Apply** button from **Surface Mesh** dialog box to accept selection. Part mesh is generated and highlighted in white. When system prompts “**Accept Mesh?**”, select **Yes** button. Compare created part mesh with Figure 2.9.

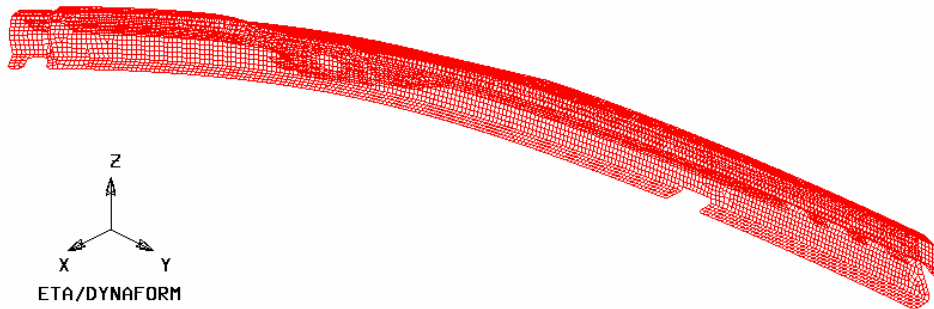


Figure 2.9: Part mesh

8. Select **Exit** button from **Surface Mesh** dialog box to complete the operation.
9. Now, you may toggle off the checkbox of **Surface** and **Lines** options from **Display Options** (shown in Figure 2.10) at the lower right bottom corner of the screen to hide all lines and surfaces.

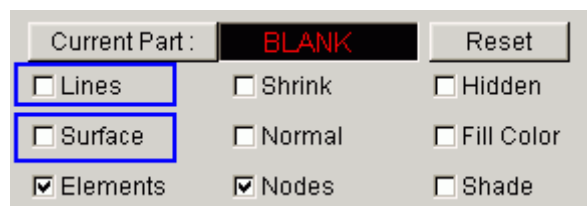


Figure 2.10: Display options

V. MESH CHECK

Inferior meshes may cause problems in stamping simulation. Therefore, the mesh quality should be checked. You continue to select the **MESH REPAIR** function (shown in Figure 2.11) to check and repair inferior mesh. The **Model Check & Repair** dialog box shown in Figure 2.12 is displayed.

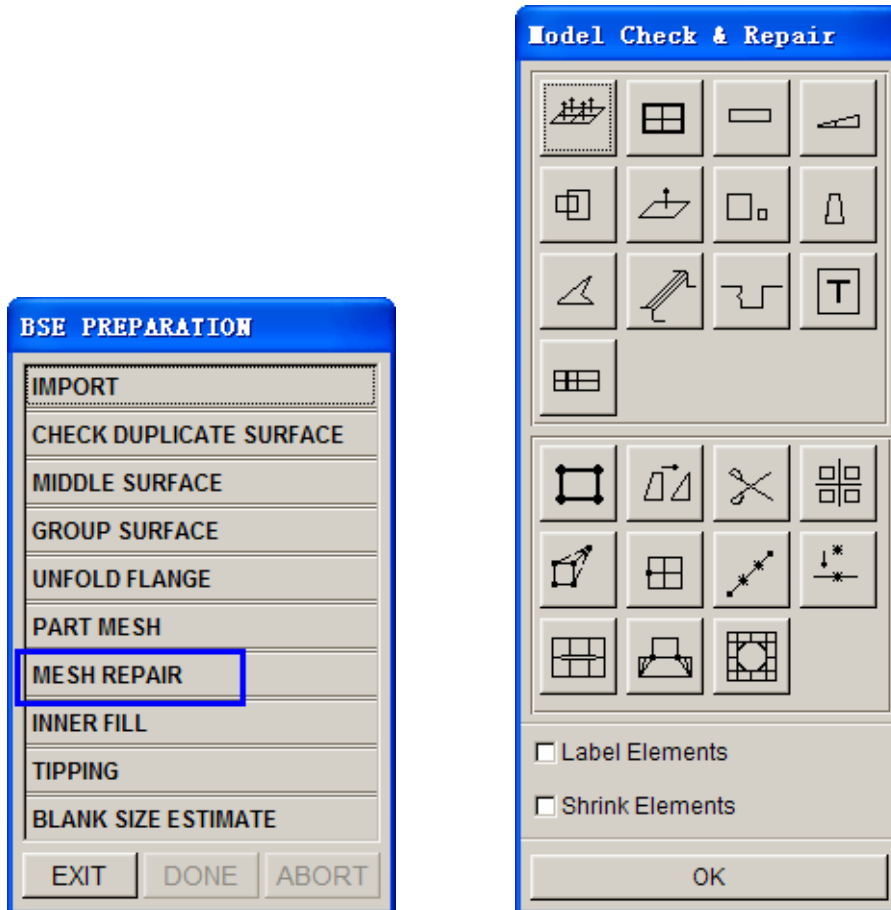


Figure 2.11: BSE preparation dialog box Figure 2.12: Model check & repair dialog box

As shown in Figure 2.12, the **Model Check & Repair** dialog box provides some useful functions that help you to check mesh quality and repair inferior mesh. In this example, you will check element warpage angle, boundary, and normal direction.

Checking Element Warpage Angle



1. Click the **Warpage** icon to display the Input dialog box shown in Figure 2.13.
2. In the input field, key in the criteria for warpage angle of **3.0°**.

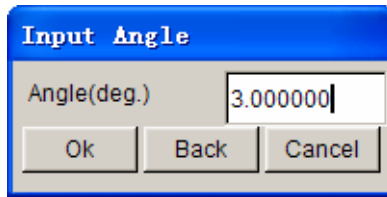


Figure 2.13: BSE preparation dialog box

3. Select **Ok** button to initiate warpage angle inspection. If warpage angle of elements exceed the criteria, eta/DYNAFORM will highlight these elements and pop up the dialog box shown in Figure 2.14. The number of failed elements is also printed in the message prompt window.

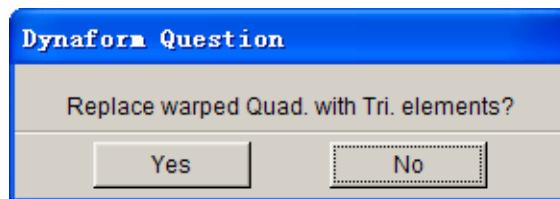


Figure 2.14: Dynaform question dialog box

4. Select **Yes** button to replace the disqualified quadrilateral elements with triangular elements. If you select **No**, the dialog box shown in Figure 2.15 is displayed. You may choose to keep the failed elements in current or new part.

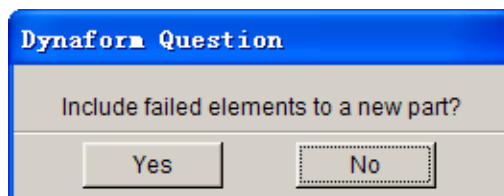


Figure 2.15: Dynaform question dialog box

5. Click **EXIT** button to quit the operation.



This function is utilized to check gaps, holes, degenerated elements, and displays defected elements with highlighted boundary.

1. Click the **Boundary Display** icon. Then, observe the displayed model. Your display should resemble Figure 2.16.

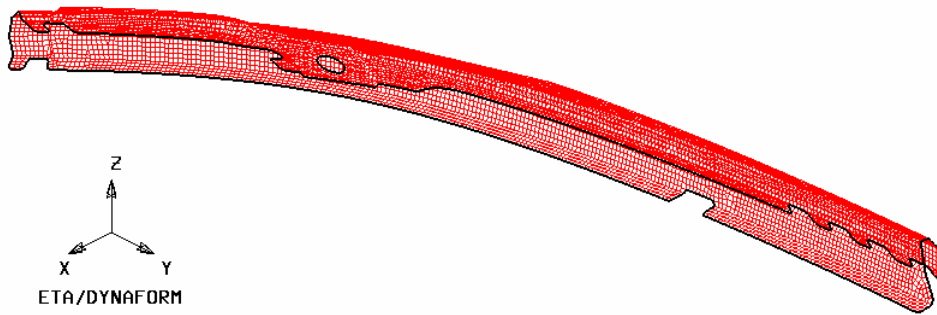



Figure 2.16: Boundary of part

- To clearly view the boundary, toggle off the checkbox of **Elements** and **Nodes** options in the **Display Options** dialog at the bottom right corner of the screen. It will help you to easily locate the tiny gaps and holes.

- Now, click on the  icon to rotate the boundary line illustrated in Figure 2.17. Examine boundary line for tiny and/large white dots. If non are found, the part mesh is free of disqualified elements. You may skip the mesh repair operation.

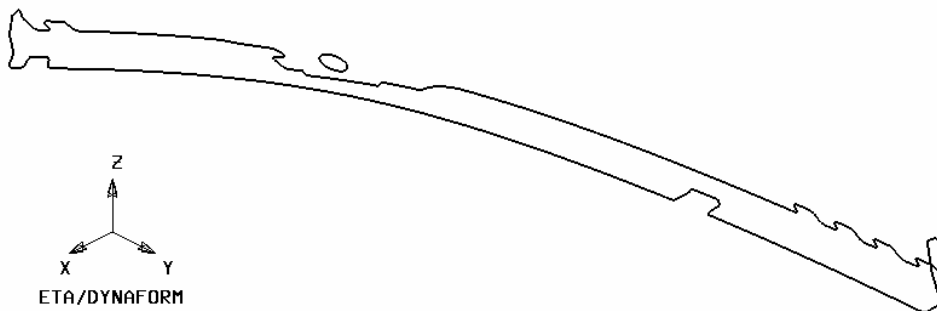



Figure 2.17: Dynaform question dialog box

- Use other checking functions, check and delete element with too small size and overlap elements.
- Click on  icon in the **Icon bar** to refresh the screen.
- Toggle on the checkbox of **Elements** and **Nodes** options in the **Display options** dialog.

Auto Plate Normal



1. Select **Auto Plate Normal** icon to display the **Control key** dialog box.
2. This dialog box provides two options: check all active parts and cursor pick part. The default setting is to check all active parts. Use your mouse cursor to pick the “**Cursor pick part**” option. Then, pick any element on the part using your mouse cursor.
3. An arrow displayed on the screen indicates the normal direction of the selected element. A popped up dialog box prompts “**Is normal direction acceptable?**”. See Figure 2.18.

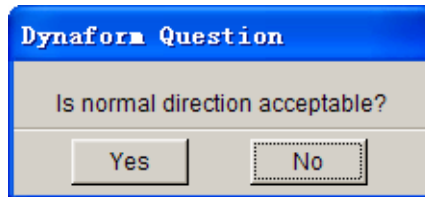


Figure 2.18: Dynaform question dialog box

4. Select **Yes** adjust element normals according to the displayed direction. If you select **No**, the element normals will be reversed.
5. Click **Exit** button to quit the operation, following by clicking **OK** button to dismiss the Model Check & Repair dialog box.
6. Exit the **BSE preparation** dialog box. Next, save your database.

VI. MSTEP MODULE AND PARAMETER SETUP

1. Select **BSE**→ **MSTEP** (shown in Figure 2.19) to enter the MSTEP GUI.

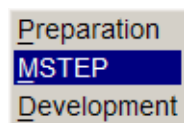


Figure 2.19: BSE menu

2. The MSTEP GUI is illustrated in Figure 2.20.

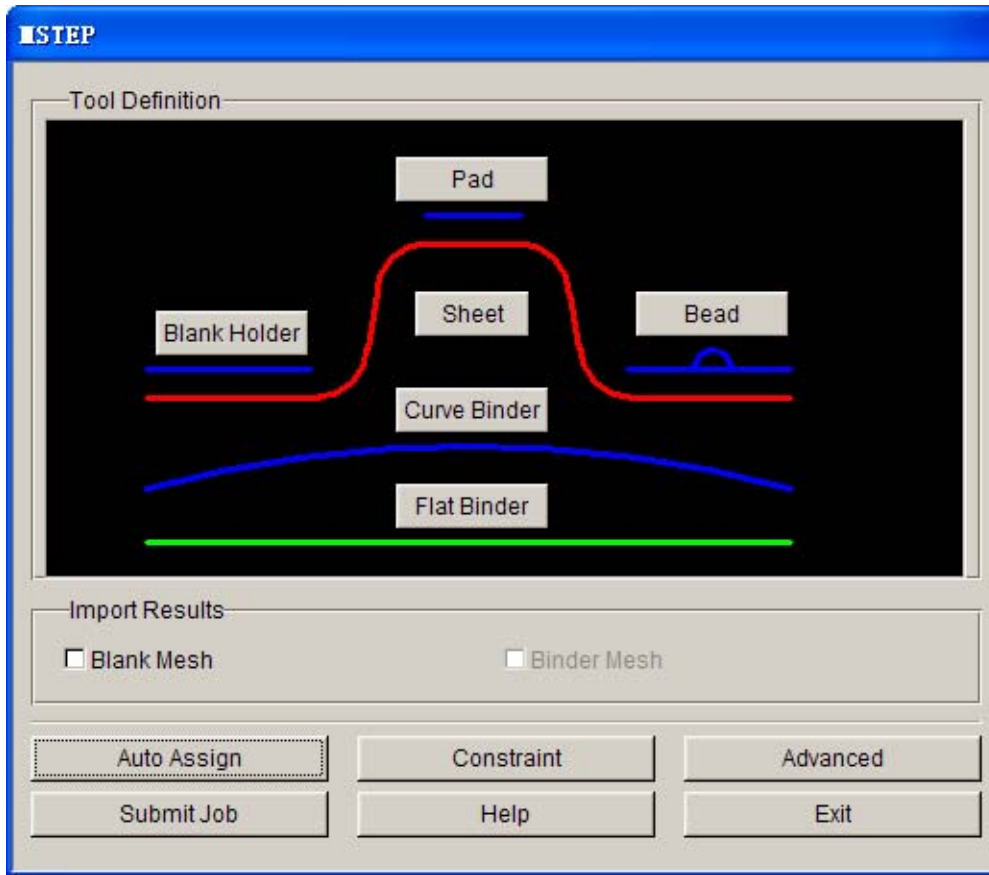


Figure 2.20: MSTEP GUI

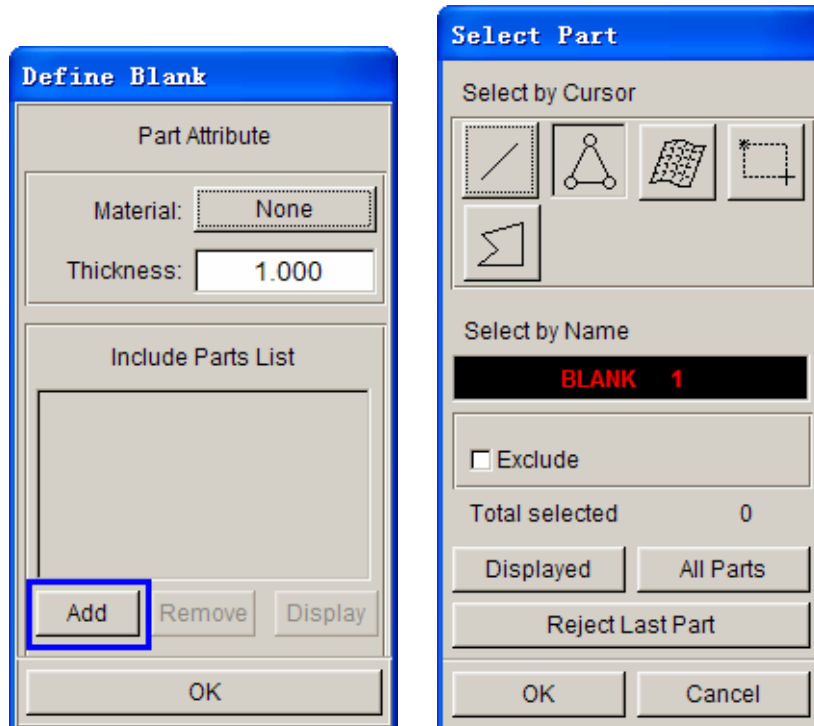


Figure 2.21: Define blank dialog box Figure 2.22: Select part dialog box

3. Define Tool

- 1) Select **Sheet** button in **MSTEP** interface to display the **Define Blank** dialog box illustrated in Figure 2.21.
- 2) Select Add button to display the **Select Part** dialog box illustrated in Figure 2.22. Use your mouse cursor to pick the **BLANK** part as the Sheet. You will observe all elements are highlighted.
- 3) Click **OK** button to accept selection and return to **Define Blank** dialog box. The selected **BLANK** part is added to the Include Part List. See Figure 2.23.

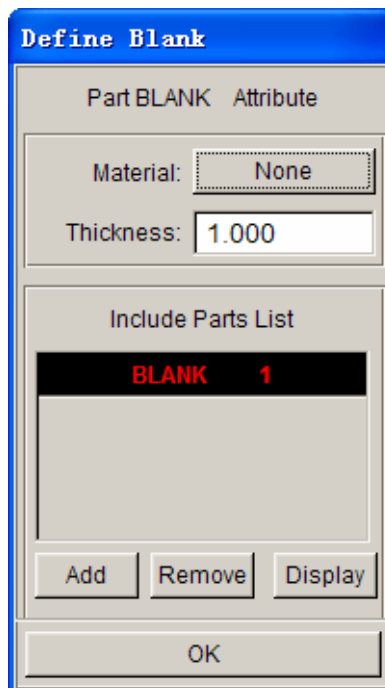


Figure 2.23: Define blank dialog box

- 4) Click on the **None** button next to Material field to display the **Material** dialog box illustrated in Figure 2.24. Again, you observe all elements in the display area are highlighted.
- 5) Select the Material Standard as **UNITED STATES**. See Figure 2.24
- 6) Then, click on the **Material Library** button in Material dialog box to display the Material Library window illustrated in Figure 2.25. Select Mild Steel **DQSK Type 36** as material for the part **BLANK**.

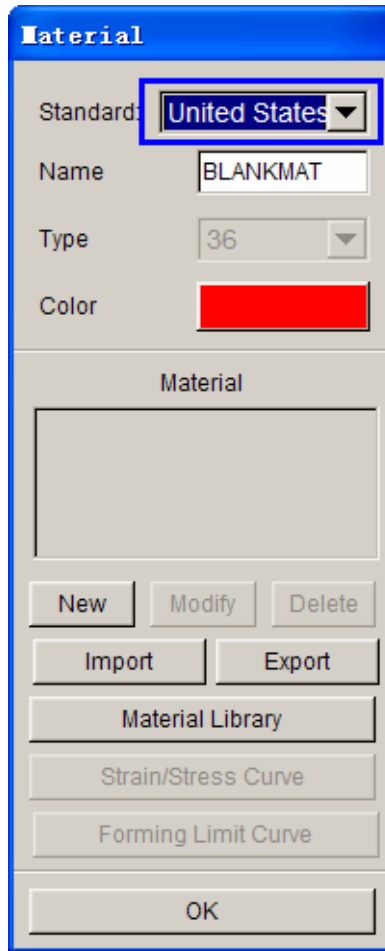


Figure 2.24: Material dialog box

Dynaform Material Library		Type 1	Type 18	Type 24	Type 36	Type 37	Type 39	Type 64
Strength Level	Material Name	ELASTIC	POWER	LINEAR	3-PARAM	ANISOTR	FLD_TRA	RATE_SEN
STEEL	Mild	CQ	+	+	+	+	-	-
		DQ	+	+	+	+	-	-
		DQSK	+	+	+	+	-	-
		DDQ	+	+	+	+	-	-
	Medium	BH180	+	+	+	+	+	-
		BH210	+	+	+	+	+	-
		BH250	+	+	+	+	+	-
		BH280	+	+	+	+	+	-
	High	HSLA250	+	+	+	+	+	-
		HSLA300	+	+	+	+	-	-
		HSLA350	+	+	+	+	+	-
		HSLA420	+	+	+	+	+	-
	Advanced High	DP500	+	+	+	+	+	-
		DP600	+	+	+	+	+	-
	Hot Rolled	CQ	+	+	+	+	+	-
		DQSK	+	+	+	+	+	-
		DDQIF	+	+	+	+	+	-
		HSLA400	+	+	+	+	+	-
	Stainless	SS11CrCb	+	+	+	+	+	-
		SS18CrCb	+	+	+	+	+	-
SS304		+	+	+	+	+	-	
SS409Ni		+	+	+	+	+	-	
ALUMINUM	AA5182	+	+	+	+	+	-	
	AA5454	+	+	+	+	+	-	
	AA5754	+	+	+	+	+	-	
	AA6009	+	+	+	+	+	-	

Figure 2.25: Material library window

- 7) Click **OK** button to return to Material dialog box. The selected material type in the last operation is added to the Material list (shown in Figure 2.26).

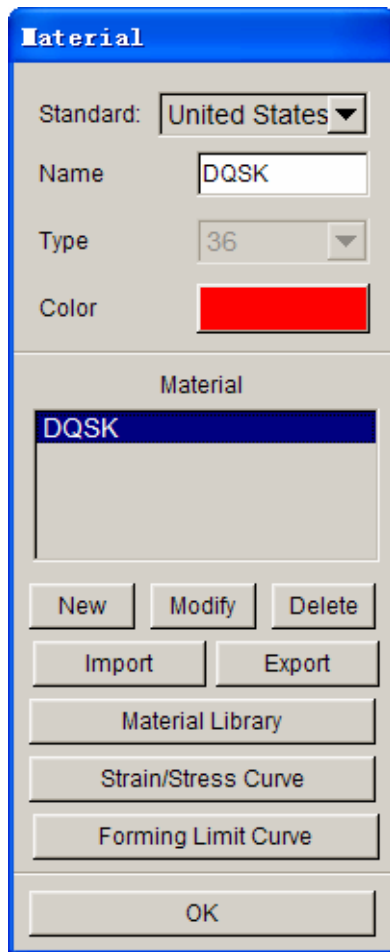


Figure 2.26: Material dialog box

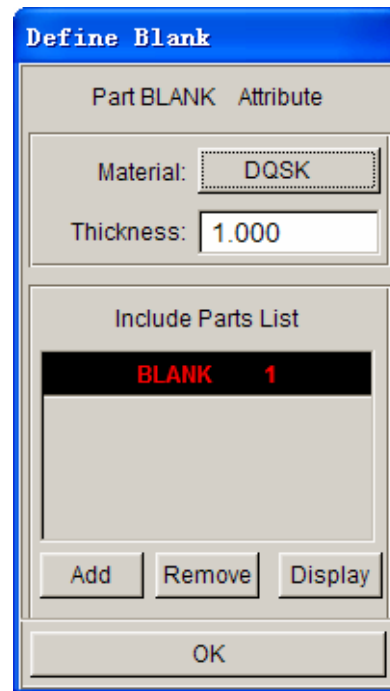


Figure 2.27: Define blank dialog box

- 8) Click **OK** button to return to Define Blank dialog box. Now, you observe the **None** button next to the Material field is changed to **DQSK**, indicating the material type is assigned to the part.
- 9) Keep the default blank thickness as **1.00** (mm).
- 10) Now, the relevant parameters for blank are defined. Click **OK** button in Define Blank dialog box to return to MSTEP GUI.
- 11) Observe the color of **Sheet** is changed from red to green, indicating definition of sheet is complete. See Figure 2.28.

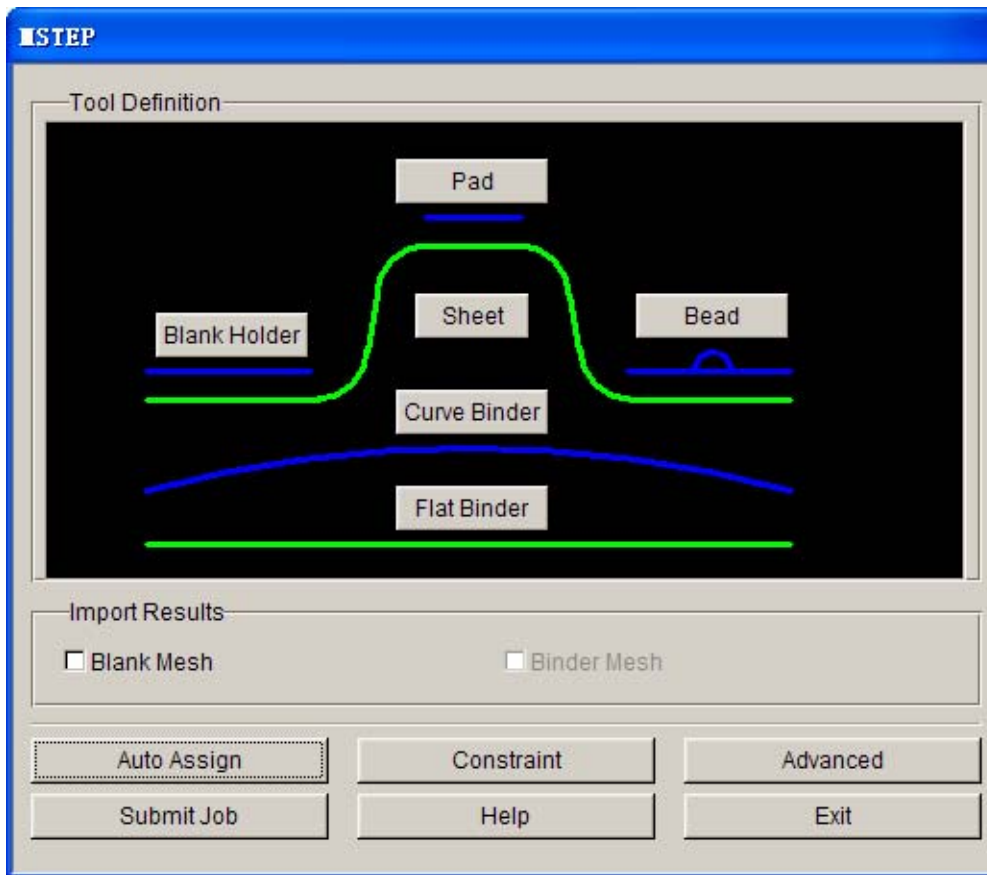


Figure 2.28: MSTEP GUI after sheet definition

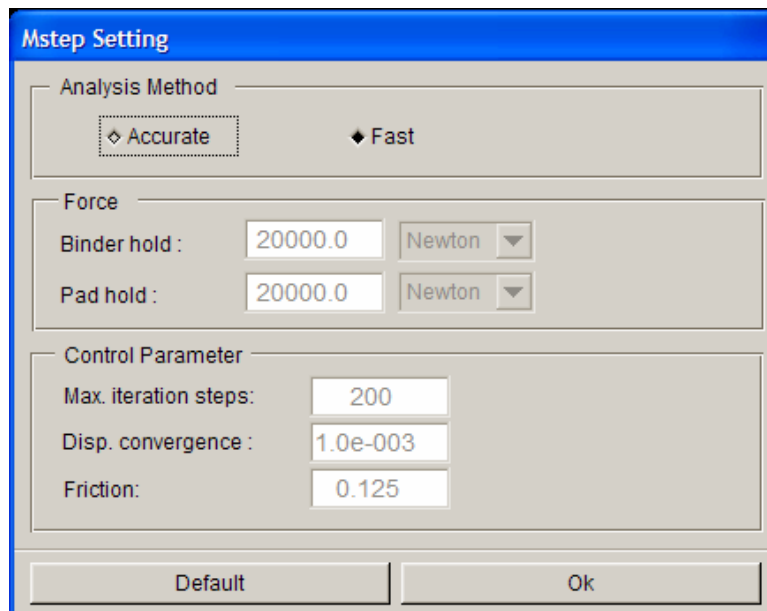


Figure 2.29: MSTEP Setting dialog box

4. Define Simulation Parameter

- 1) Click on the **Advanced** button in MSTEP GUI to display the **MSTEP SETTING** dialog box shown

in Figure 2.29.

- 2) Two solver options are provided: **Accurate** and **Fast**. The **Accurate** option enables advanced simulation which considers blank holder pressure, pad pressure, and draw bead infection, together with material parameter and plasticity behavior of material. It leads to more accurate calculation result. It is suitable for evaluation of conceptual tool design by checking product formability, getting blank outline, determine process planning and estimate effect of process parameters on the forming process. The **Fast** option facilitates quick and effectively blank unfolding for material cost estimation. There is no consideration of the effect of real process parameters such as blank holder pressure, pad pressure and draw bead pressure.

In this example, the **Fast** option is selected.

- 3) Click **OK** button to return to MSTEP GUI.

5. Start the MSTEP Solver

Now, all relevant parameters are defined. You can proceed to running the simulation by clicking on the **Submit Job** button in MSTEP GUI.

VII. START UP POST-PROCESSOR AND ANALYZE SIMULATION RESULT

After MSTEP calculation is complete, the unfolded blank outline is displayed in the display area. See Figure 2.30.

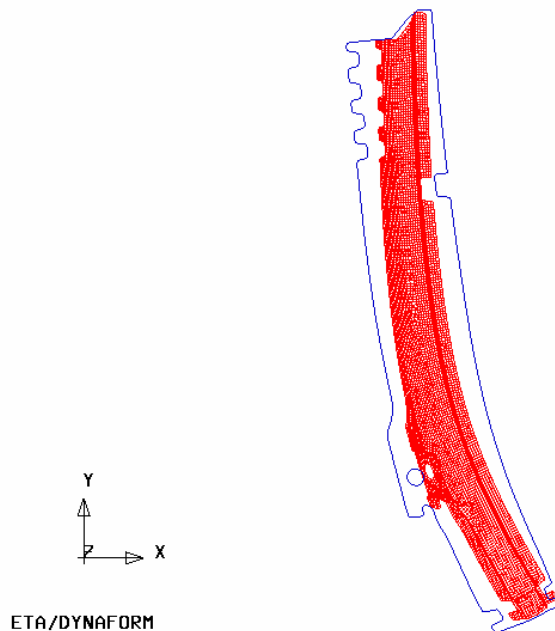



Figure 2.30: Blank outline

In order to view detailed information, you may use the post processor to analyze result file.

1. Select **PostProcess** from the **Menu** bar to open eta/POST. The eta/POST interface is displayed.

2. Select **File**→**Open** (illustrated in Figure 2.31) or  icon

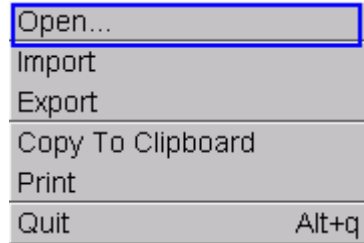


Figure 2.31: File manager

3. From the popped up window, pick the **dynain.mstep** file using your mouse cursor. Then, click **Open** button, read in the result file. The part illustrated in Figure 2.32 is shown on the displayed area.



Figure 2.32: Part shown in display area

4. Forming Limit Diagram

- 1) Select **FLD** icon from the **Special** icon bar illustrated in Figure 2.33.
- 2) Select **Middle** of the **Current Component** pull down menu. See Figure 2.34.
- 3) Click **FLD Curve Option** button to set FLD parameters (n, t, r, etc)
- 4) Select **Edit FLD Window** button to locate position of FLD.
- 5) Click **PLOT** button to display the distribution of FLD. See Figure 2.35.



Figure 2.33: Special icon bar for forming analysis

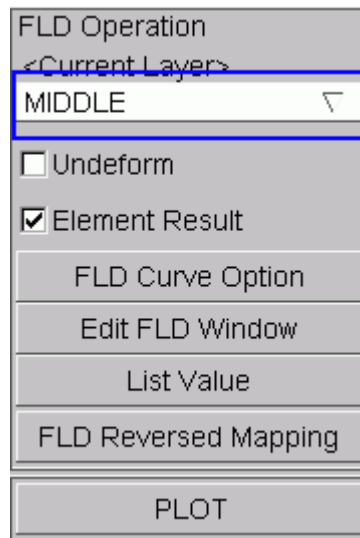


Figure 2.34: FLD dialog

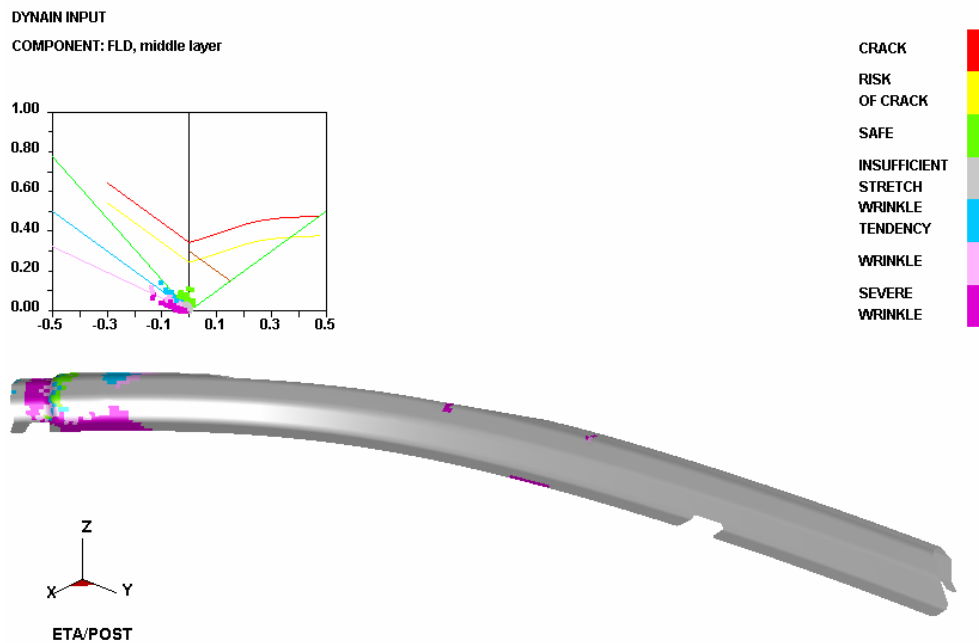


Figure 2.35: FLD distribution

5. Thickness change/ thinning change.

- 1) Select **Thickness** icon in the Special icon bar. See Figure 2.36.
- 2) You may select either **THICKNESS** (absolute value) or **THINNING** (relative) in the **Current Component** pull down menu illustrated in Figure 2.37.
- 3) Click **PLOT** button to display the thickness/thinning contour illustrated in Figure 2.38.



Figure 2.36: Special icon bar for forming analysis

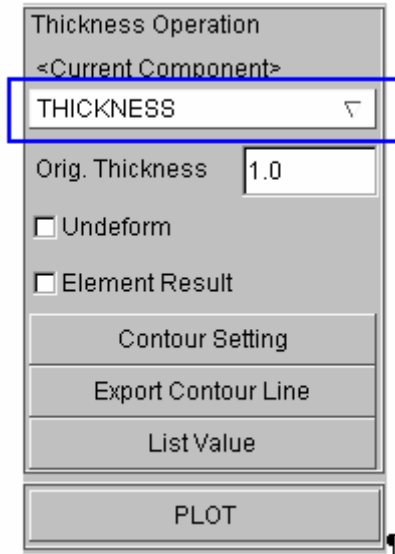


Figure 2.37: Thickness operation dialog

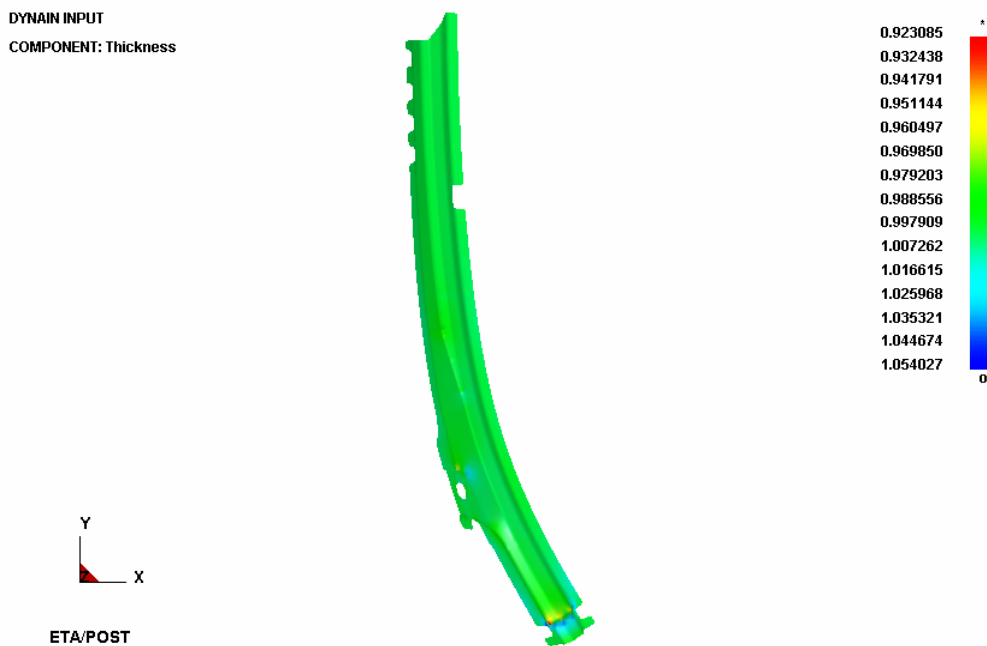


Figure 2.38: Thickness/thinning contour display

6. Import blank outline

- 1) Select **File**→**Import** from the **Menu bar**.
- 2) Use your mouse cursor to pick **example1_mstep.lin** file. Then, click on **Open** button to read in blank

outline. See Figure 2.39.

- 3) Close the eta/POST interface to return to eta/DYNAFORM interface.

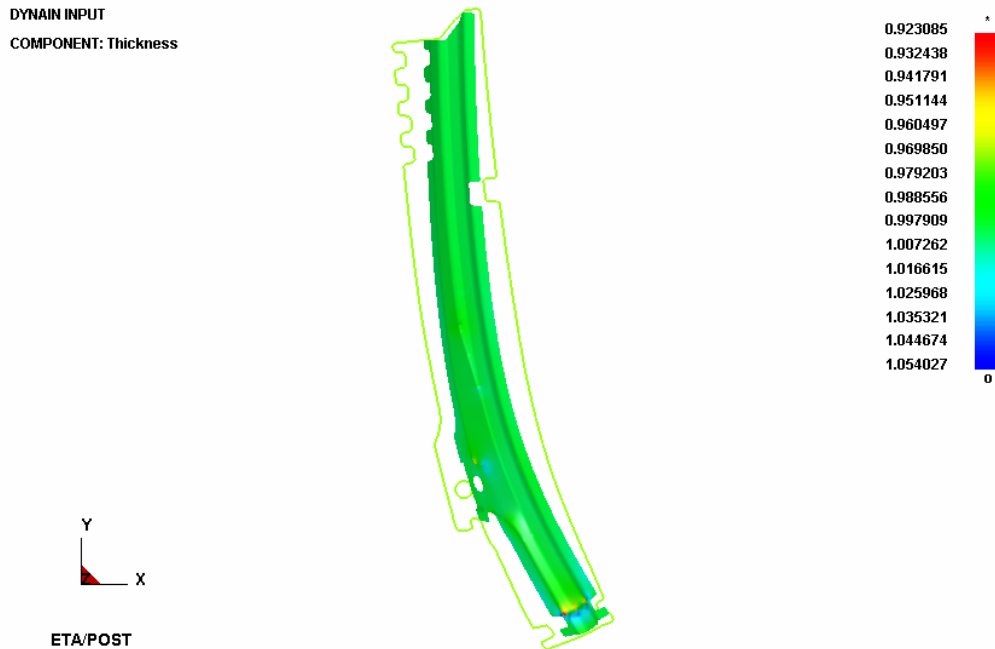


Figure 2.39: Blank outline

VIII. Blank Nesting

1. Select **BSE**→**Development** to display the BSE Development dialog box illustrated in Figure 2.40.
2. From the dialog box, select **BLANK NESTING** function to display the **Blank Nesting** dialog box illustrated in Figure 2.41. Refer to eta/DYNAFORM User's Manual for detailed description of functions provided in the Blank Nesting dialog box.
3. Click on the **Blank Outline (Undefined)** button to select the blank outline for nesting calculation. The Select Line dialog box is displayed.
4. Use your mouse cursor to pick the blank outlines. Click **Ok** button to confirm the selection.

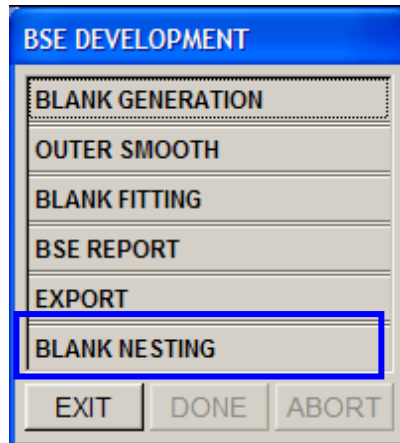


Figure 2.40: BSE development dialog box



Figure 2.41: Blank nesting dialog box Figure 2.42: Blank nesting dialog box after line selection

5. You observe the **Blank Outline (Undefined)** button is changed to **Blank Outline** button. The Material and Parameters fields are also enabled.
6. Click on the **Apply** button to begin nesting calculation. The **Result** tab in **Blank Nesting** dialog box is displayed (shown in Figure 2.43). The nesting result is shown in Figure 2.44.
7. Scroll the vertical bar in **Result** page to view material utilization as result of different angle, pitch and blank width.
8. Next, click on the **Output Nest Report** button to display the **Nest Report** dialog box illustrated in Figure 2.45.
9. In the input data field of Production Volume, key in **100,000**.
10. Key in base material cost, **0.50**.
11. Click on the **Apply** button to output the nest report in HTML format, shown in Figure 2.46.
12. Close the web browser.
13. Click the **Cancel** button to dismiss **Nest Report** dialog box, following by clicking **Exit** button to dismiss **Blank Nesting** dialog box.
14. Exit Blank Development dialog box.
15. Save your database.

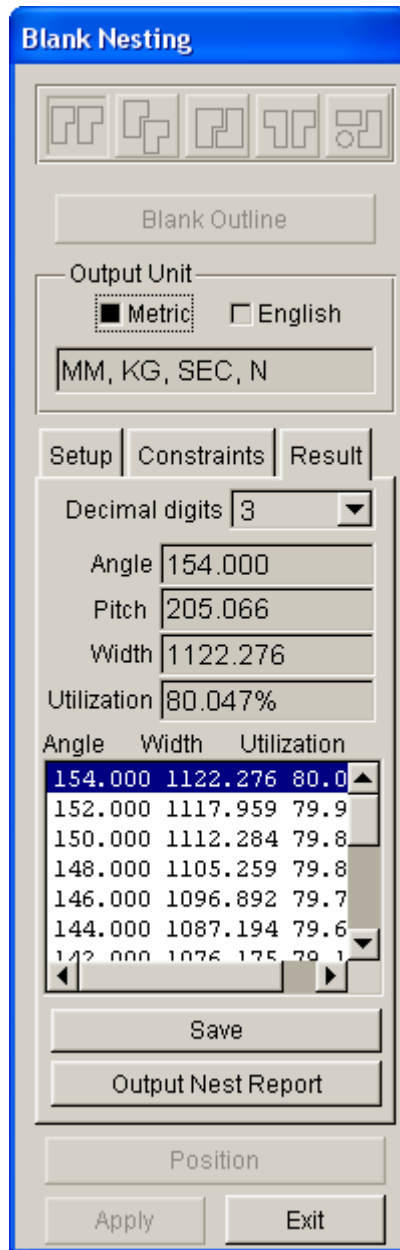


Figure 2.43: Result page

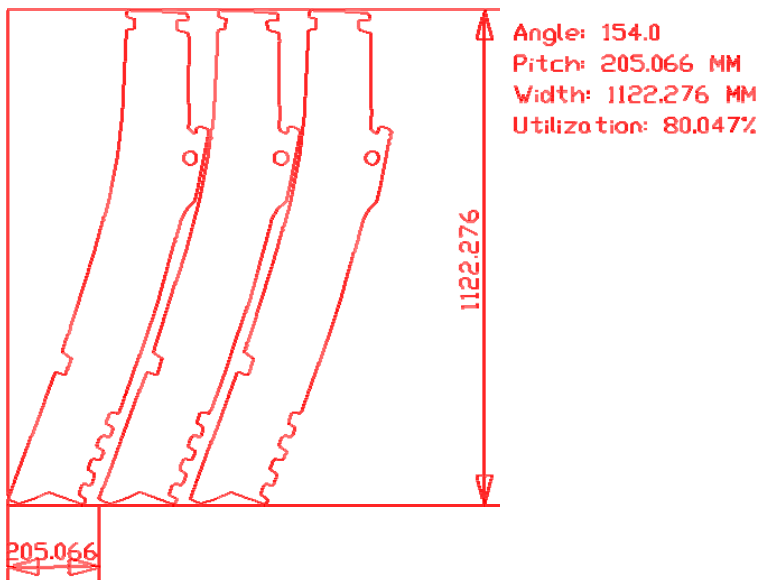


Figure 2.44: Nesting layout

Nest Report

Date:

Filename:

Blank Outline Color

Output Unit

Production Volume / Per Coil
 ▼

Production Volume

Coil Length
 ▼

Base Material Cost
 ▼

Extra Material Cost
 ▼

Scrap Value Cost
 ▼

Consumables Cost

Comments

Figure 2.45: Nest report dialog box

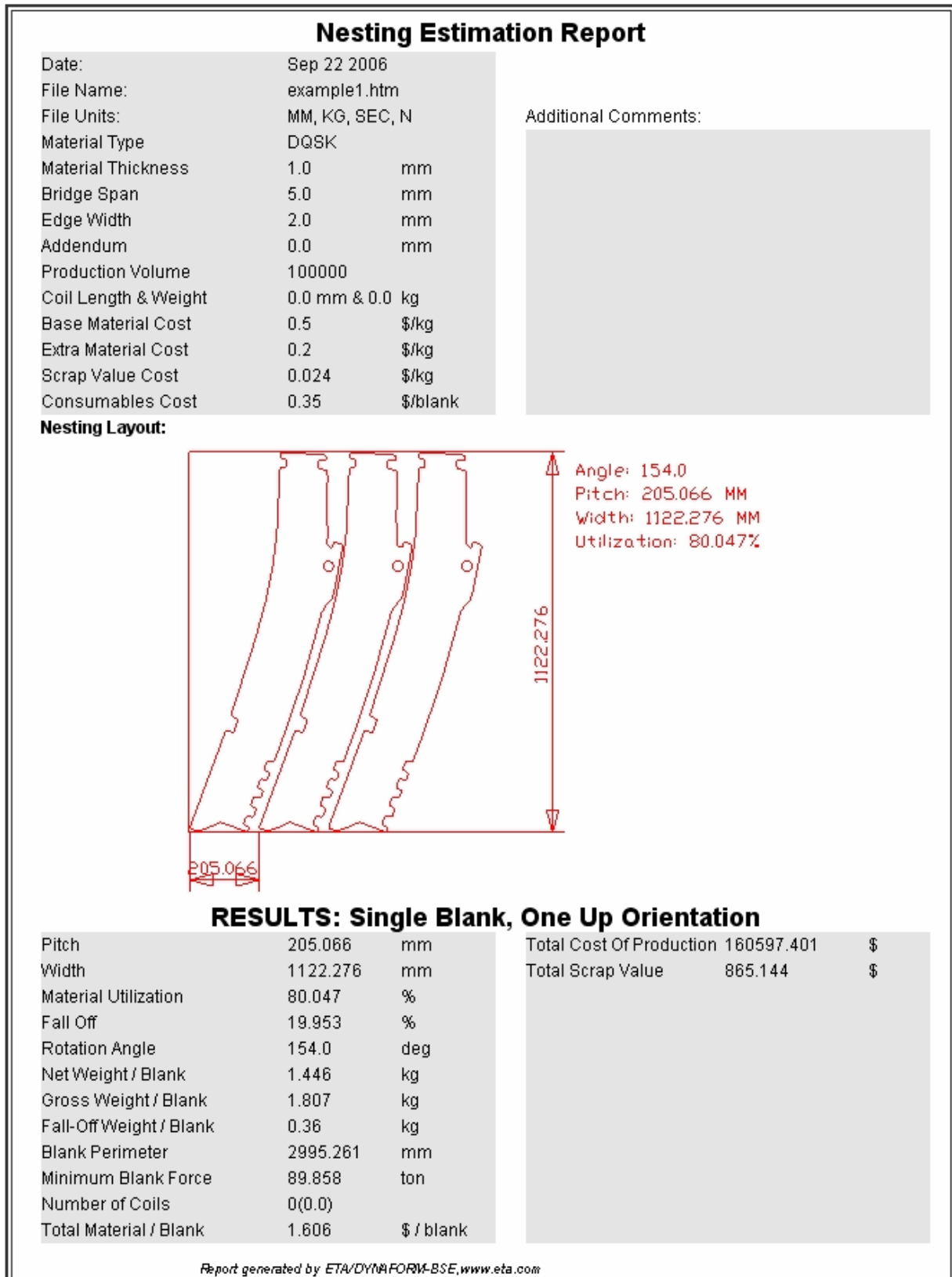


Figure 2.46: Nesting report