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.

INTRODUCTION

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, the BSE menu consists of PREPARATION, MSTEP, and DEVELOPMENT.

Figure 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 2 illustrates the streamlined process guidance graphic user interface (GUI) of *MSTEP*.

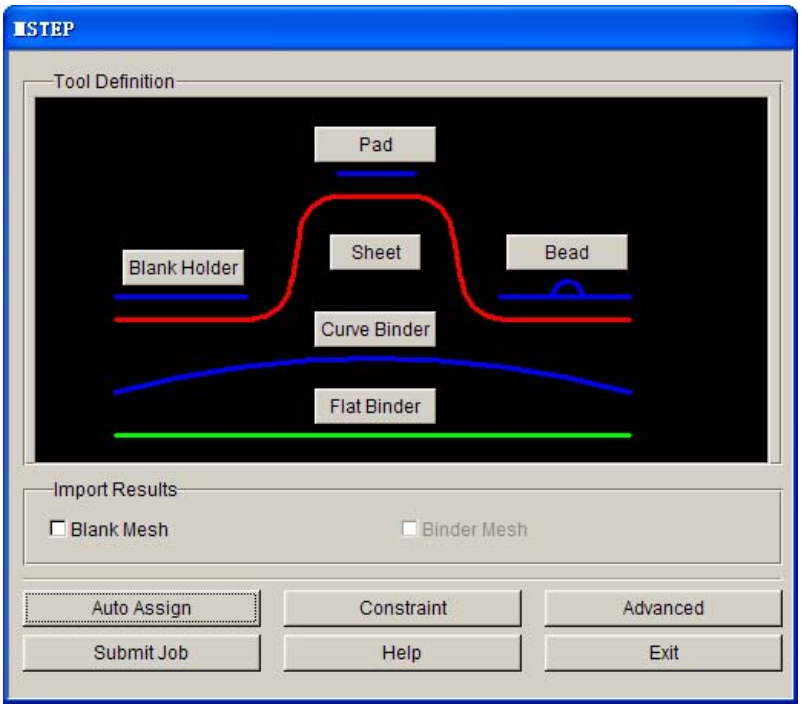


Figure 2: MSTEP GUI

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

TABLE OF CONTENTS

Example 1. Trim Line Estimation

I. Create Database and Read in Model Type

Start eta/DYNAFORM 5.5

For work shop and Linux user, enter “df55” (default) in command line to start DYNAFORM 5.5. For PC user, double click DYNAFORM 5.5 (DF55) icon or select DYNAFORM from program group to start the software.

After starting eta/DYNAFORM, program automatically create default empty database file Untitled.df. User needs to import CAD or CAE model to database to start working or to open existing database.

Open File

1. Select menu **File→Open** as shown figure 1.1or import function icon from tool bar.  .

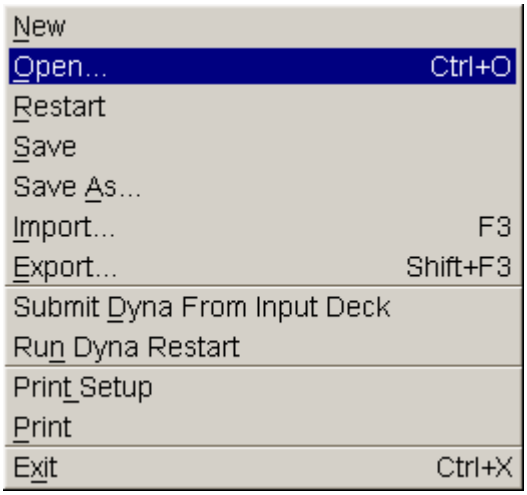


Figure 1.1: Open file menu

Find the catalogue of training input file, select model file: MSTEP_model4.df and open it.

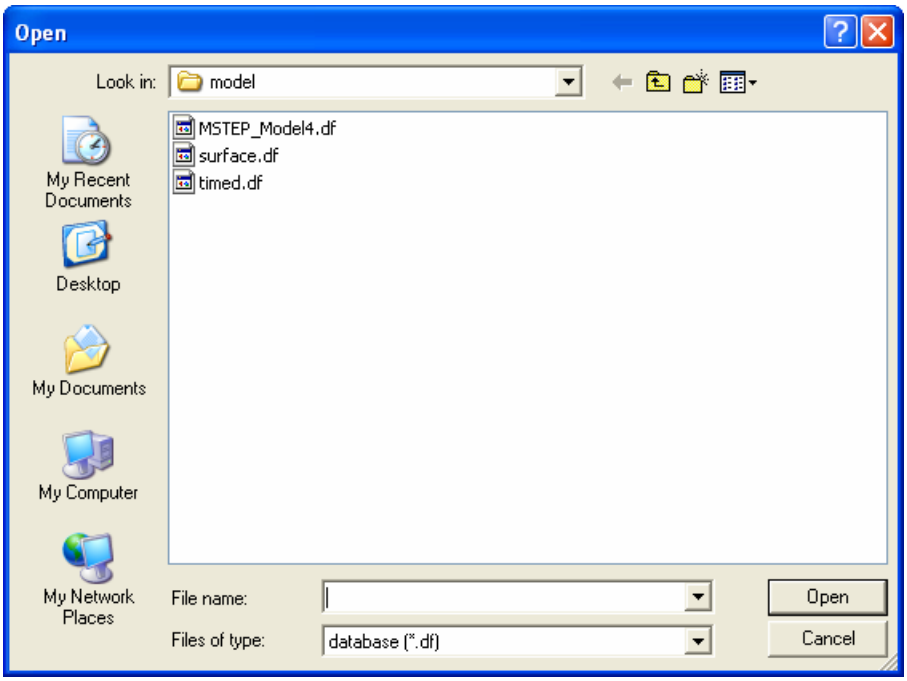


Figure 1.2: Open file window

After opening the file, verify if the model in figure below is displayed on the screen. There is plotted mesh model in the file of this example, together with products, flange, and formed CAD surface. It is displayed on the screen as a platform. See figure 1.3.

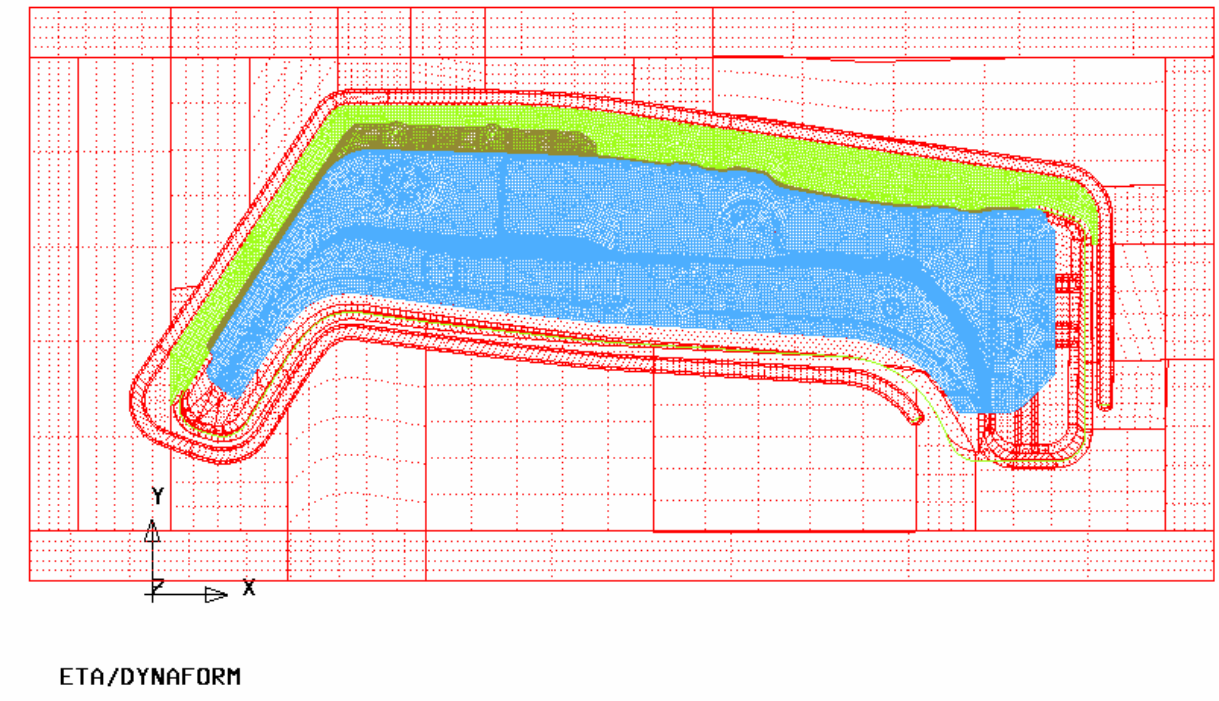


Figure 1.3: Illustration of MSTEP_model4

Note: *icons under different system platforms can be different. Functions of other icons in tool bar are introduced in the following chapters. User may also refer to eta/DYNAFORM user manual for all functions of*

tool bar.

II. Check Parts in Database

In eta/DYNAFORM, all models are managed basing on parts. In default setting, any entity is to be created or read in part. Detailed information of part operation see eta/DYNAFORM user manual.

Check part by icon  , or menu **Parts→Turn On** as shown figure1.4.

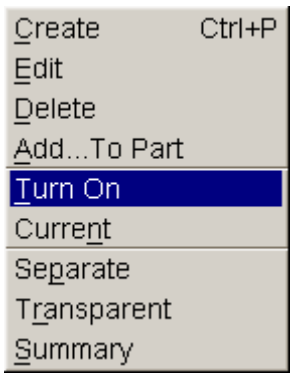
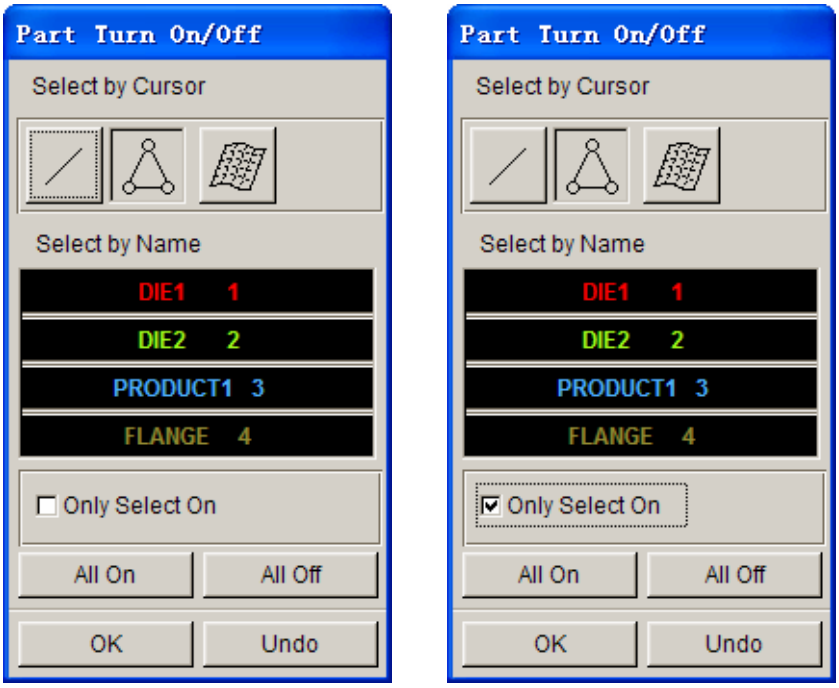


Figure 1.4: Turn on menu

1. Select **Parts→Turn On,** display open/close part dialogue box. All defined parts are displayed in the list. Parts are labeled with names and numbers. User may open and close parts using different methods.

2. Open Only Select On in the dialogue box. See figure 1.5. This means only selected part is open every time. Other parts are closed automatically.



a) Open random parts b) Only open selected part

Figure 1.5: Turn on/off dialog box

3. Complete drawing die face composed with DIE1 and DIE2. See figure 1.6~1.9

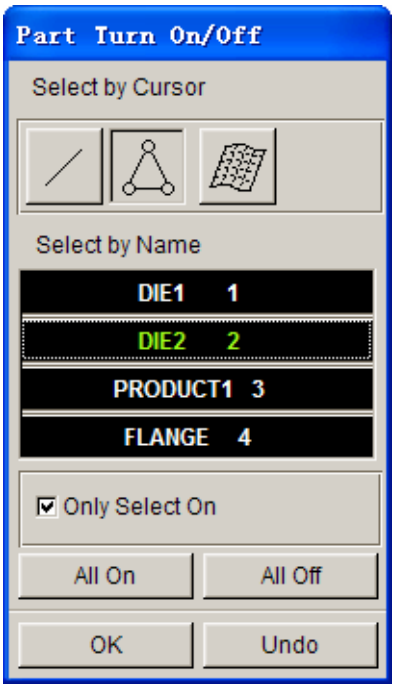


Figure 1.6: Only open DIE2

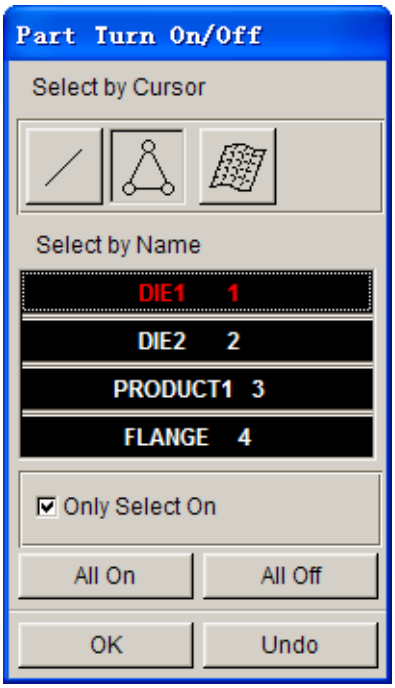


Figure 1.7: Only open DIE1

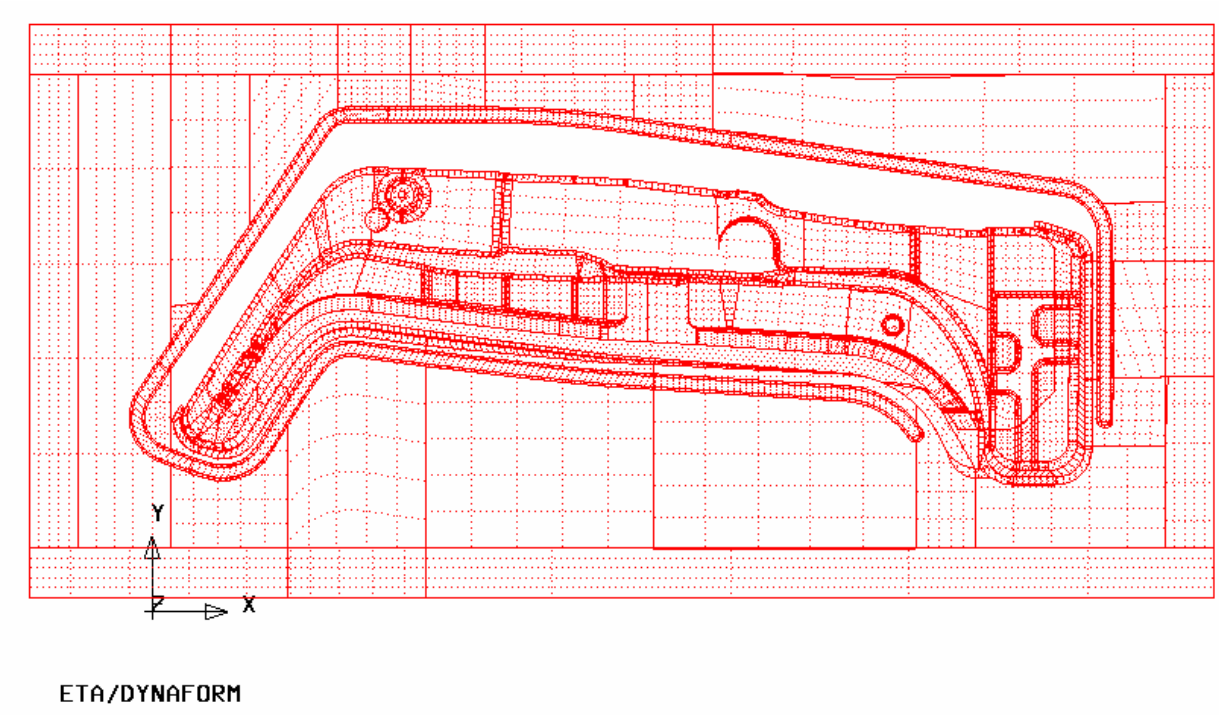


Figure 1.8: Illustration of DIE1

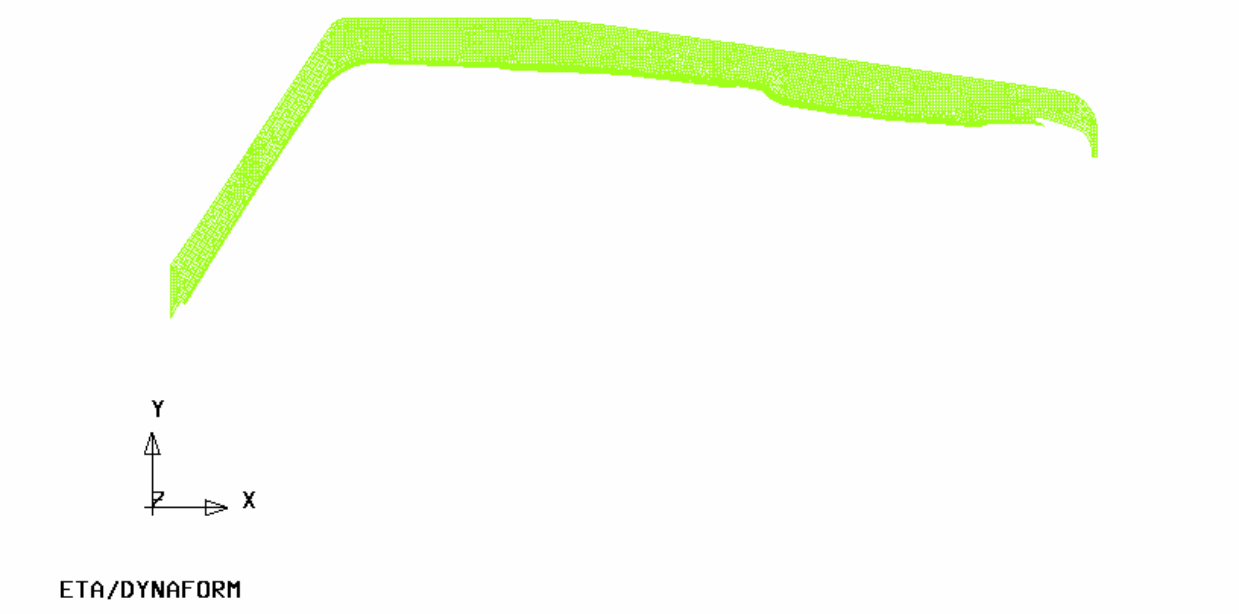


Figure 1.9: Illustration of DIE2

4. Product after flanging that is composed of PRODUCT1 and FLANGE. Part FLANGE is the flanging part. See figure 1.10~1.13.

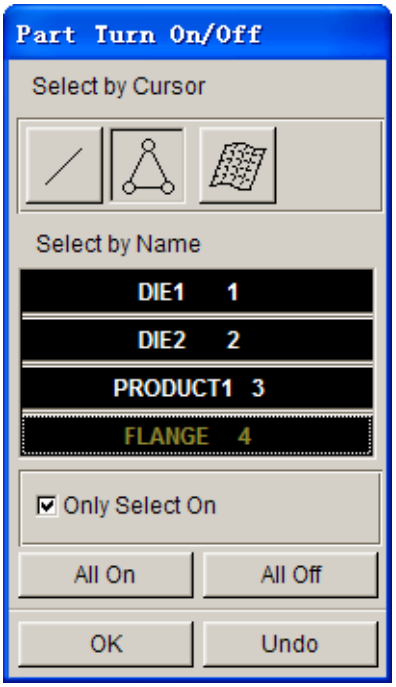


Figure 1.10: Only open FLANGE

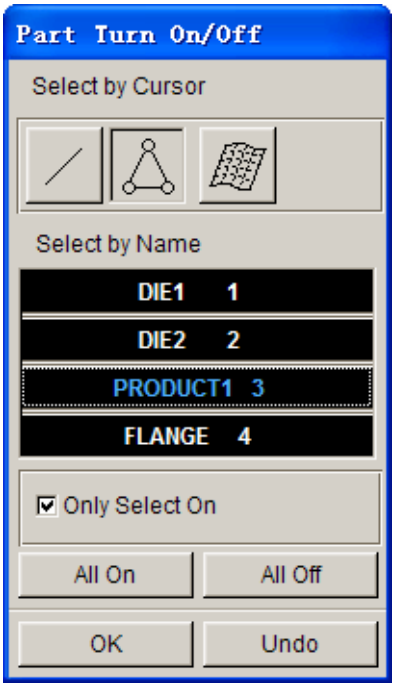


Figure 1.11: Only open PRODUCT1

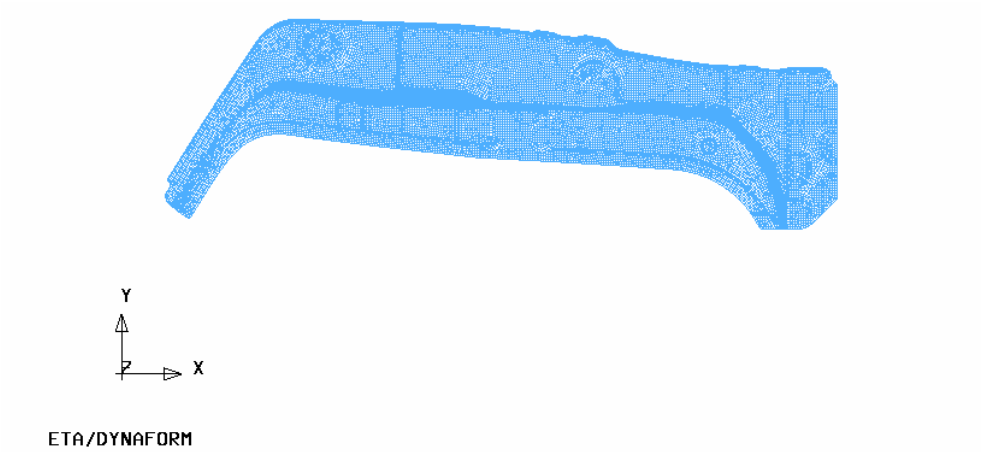


Figure 1.12: Illustration of PRODUCT1

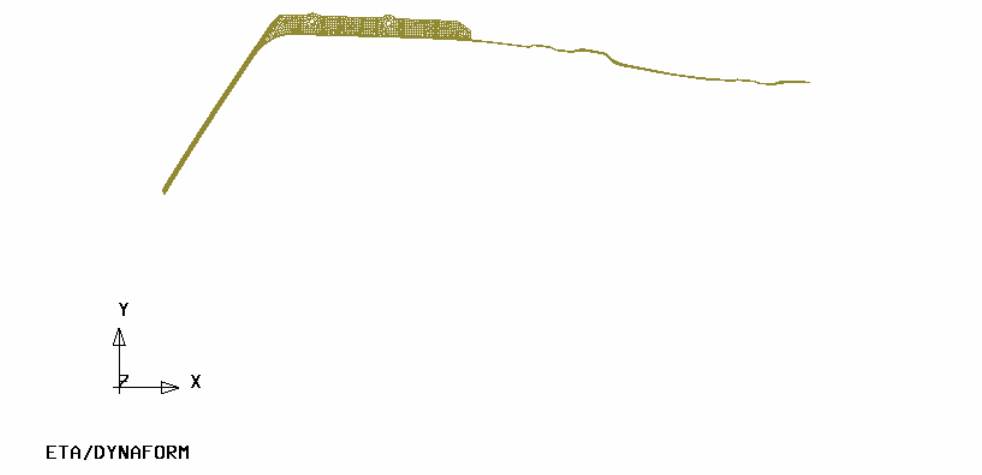


Figure 1.13: Illustration of FLANGE

III. Mesh Check

Before mesh check, open all parts and then close all surfaces as shown figure 1.14~1.15.

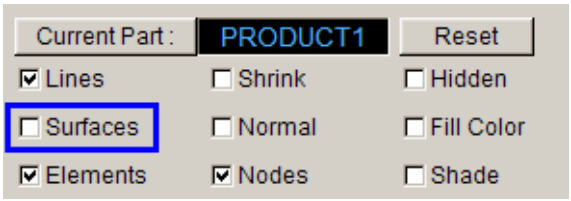


Figure 1.14: Display options

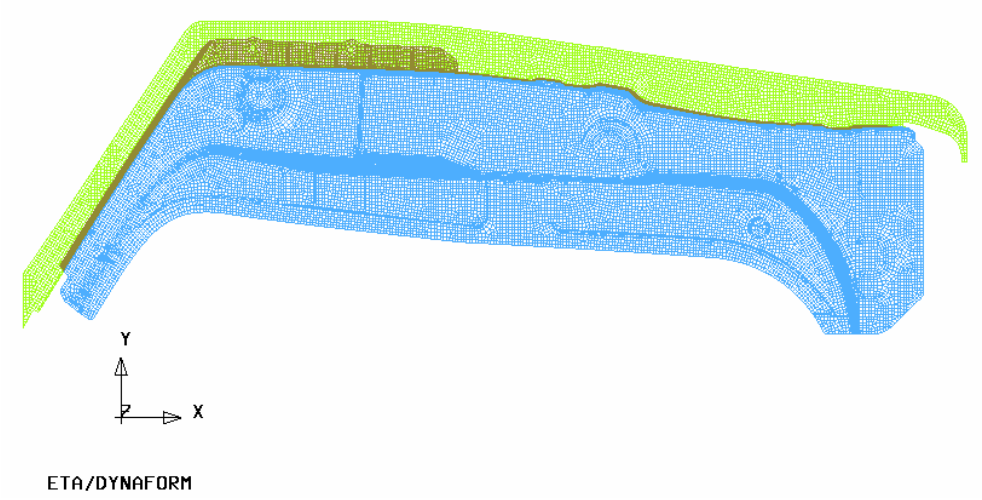


Figure 1.15: Illustration of only part meshes

In this example, the user opened file is already meshed, so it is not necessary for meshing operation. But to prevent from some potential, simulation effect defects in mesh, it is still better to check the mesh quality. All tools for checking mesh quality is in menu **Preprocess→Model Check/Repair.** Enter checking operation by selecting **Preprocess→Model Check/Repair** or through shortcut button **Ctrl+R.** See figure1.16.

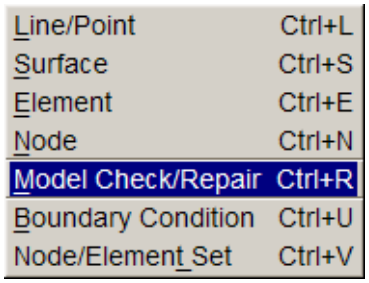


Figure 1.16: Model Check/Repair menu

Open model check dialogue box. See figure 1.17.

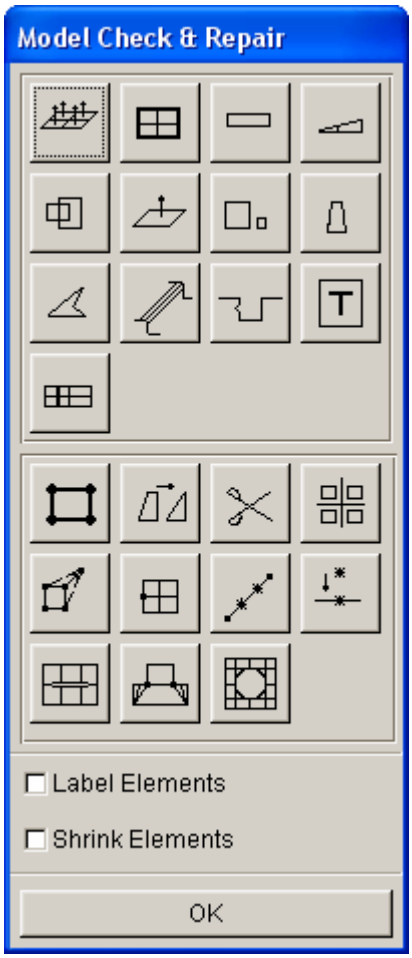


Figure 1.17: Model Check/Repair dialog box

As shown in the figure above, model check dialogue box includes some functions to help user to check mesh quality. Mainly used functions in this example are: element internal angle check, element boundary display and auto normal unification. Other functions refer to eta/DYNAFORM user manual.

Detailed checking standard of mesh model is listed as follow:

- Element internal angle > 5.0, delete or combine unqualified elements.
- Element warpage angle<3.0. Unqualified elements are automatically divided into triangle elements.
- Model boundary check is closed curve. Any existence of gaps, holes and degenerated element is not allowed in mesh model.
- All element normal are adjusted to be the same direction.

In this example, enter mesh models that match the checking standard. There is no existence of unqualified elements.

IV. MSTEP Module and Parameter Setting

1. Select menu **BSE→MSTEP** to enter MSTEP solver module as shown figure 1.18.

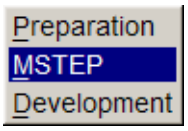


Figure 1.18: BSE menu

2. The figure 1.19 is the sketch figure of MSTEP module. The setup setting is very simple. It is only required to assign corresponding part to tool and select solving mode to execute simulation.

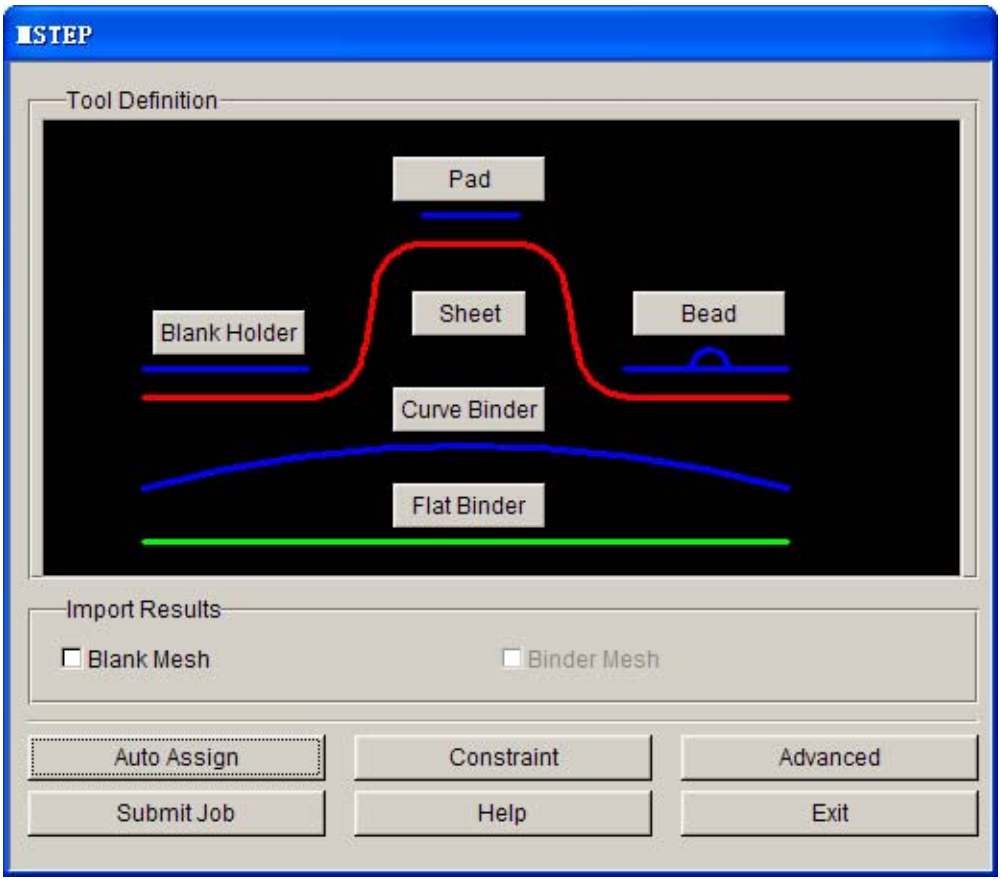


Figure 1.19: MSTEP GUI

3. Define Tool

In this example, to simplify and accelerate solving operation, part flange of the product is selected as sheet to be opened. Select part of die face (bigger than opened boundary) as the target face. This greatly reduces element models (element number), and at the same time simplify models (target surface is comparatively simple, so is sheet).

Select Sheet button in MSTEP dialogue box. Define Blanks dialogue box pops out. See figure 1.20. Select Add button to add tool to the corresponding part.

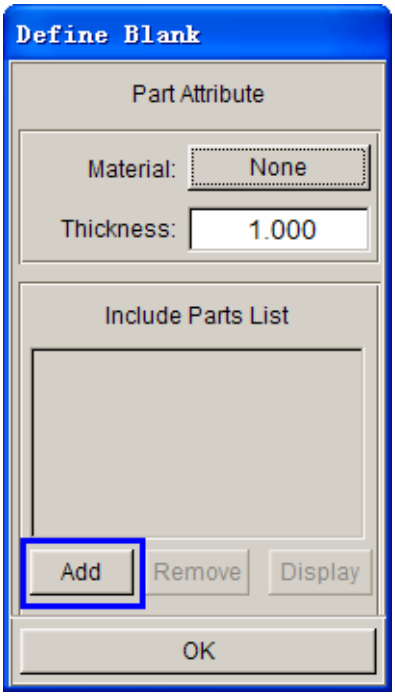


Figure 1.20: Define Blank dialog box

1) Select FLANGE part as the corresponding part of sheet tool in the Select Part dialogue box. See figure 1.21.

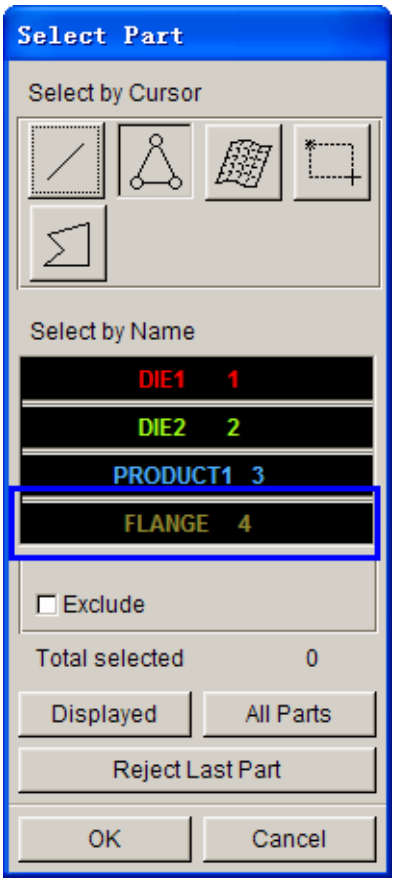


Figure 1.21: Select Part dialog box

2) Click OK button to return to Define Blank dialogue box. Selected target part is added to the Include Part List as shown figure 1.22..

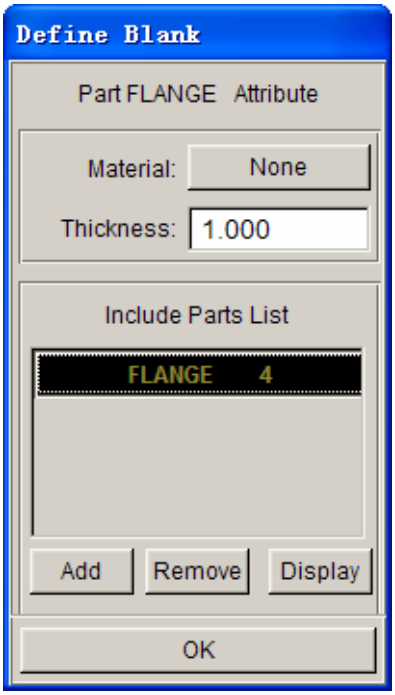


Figure 1.22: Define Blank dialog box

3) Mouse click to select FLANGE part in the list. The selected part change into black, and then select None button behind Material to enter Material dialogue box. At this time, all elements in the selected part are highlighted. See figure1.23~1.24.

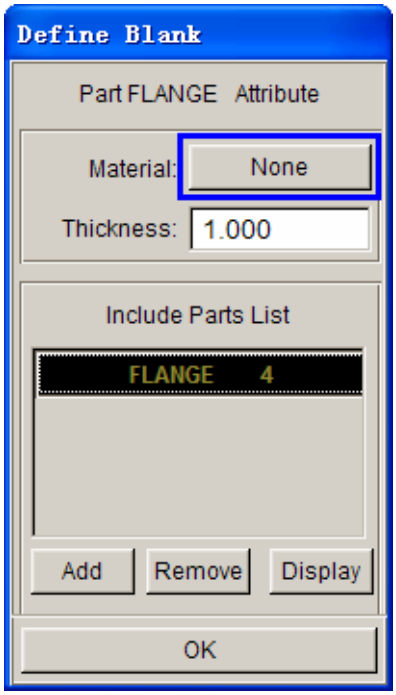


Figure 1.23: Define material

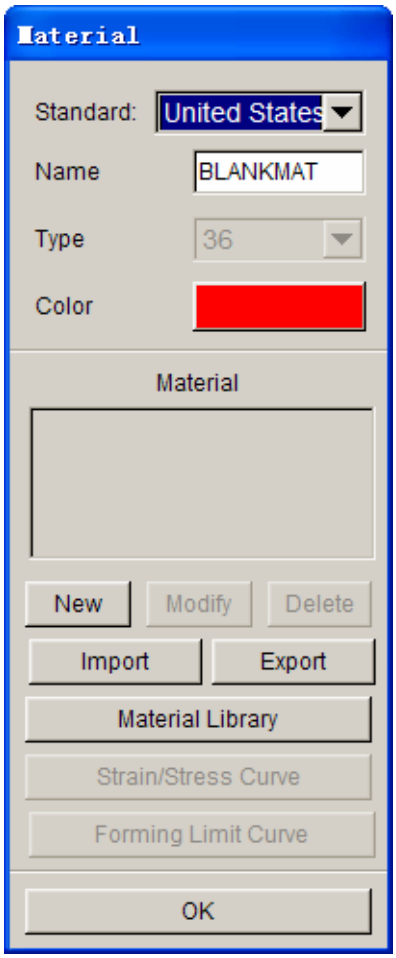


Figure 1.24: Define Material dialog box

4) Select Material Library button in Material dialogue box. Dynaform Material Library dialogue box pops out. Select DQSK 36 material as part BLANK material type. See figure1.25.

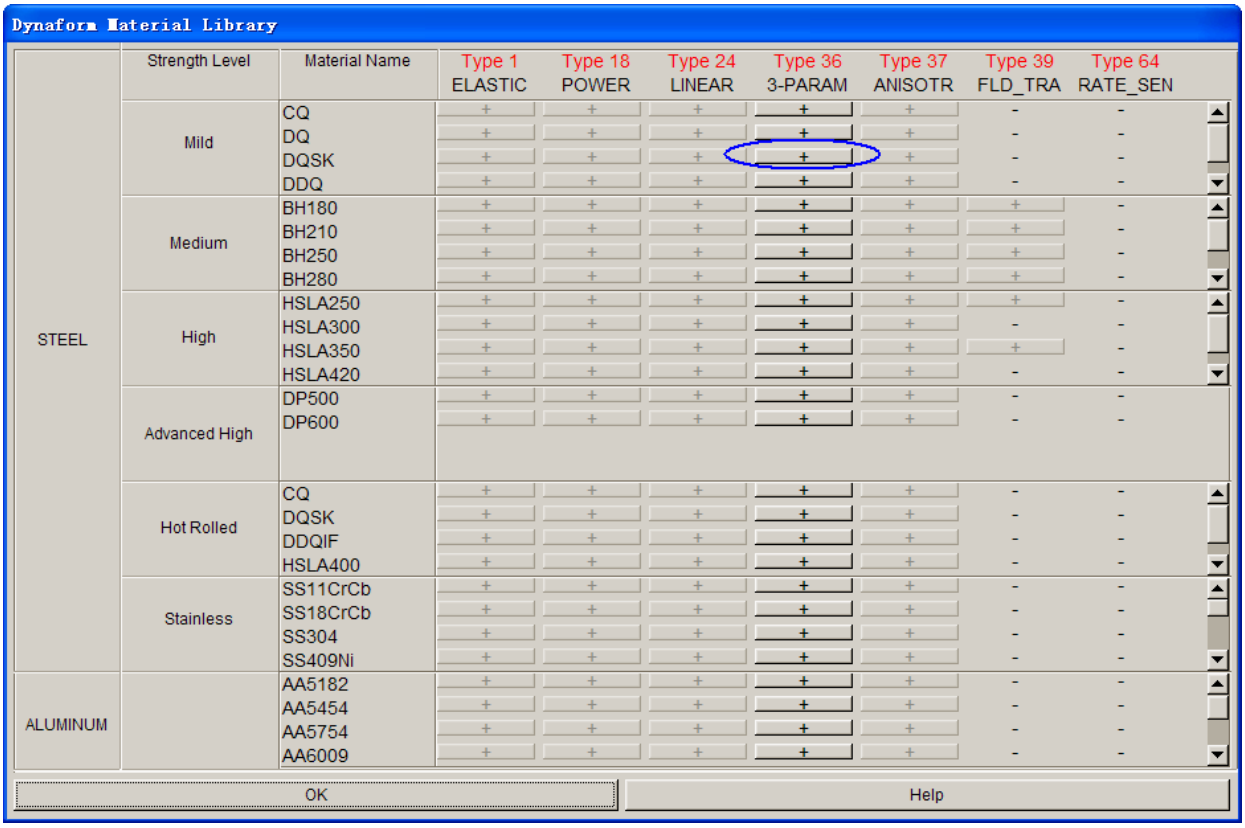


Figure 1.25: Material library window

5) Click OK to return to Material dialogue box. Selected material type in the last step is added to the Material List. See figure 1.26.

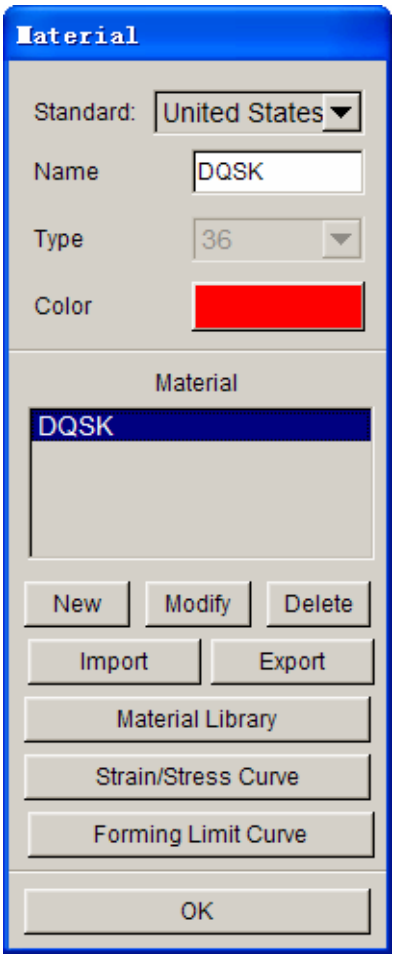


Figure 1.26: Define Material dialog box

6) Click OK button to return to Define Blank dialogue box. None button behind Material is already changed into the material type selected by DQSK, which means the part is already defined with material. See figure 1.27.

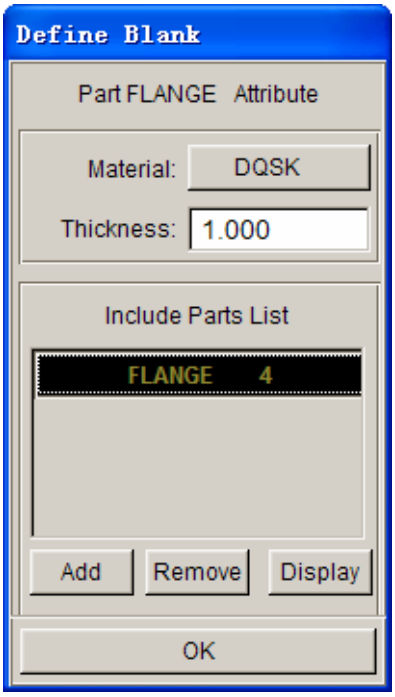


Figure 1.27: Define blank dialog box

7) Then select Thickness edit box, define blank thickness as 1.2 mm. See figure 1.28.

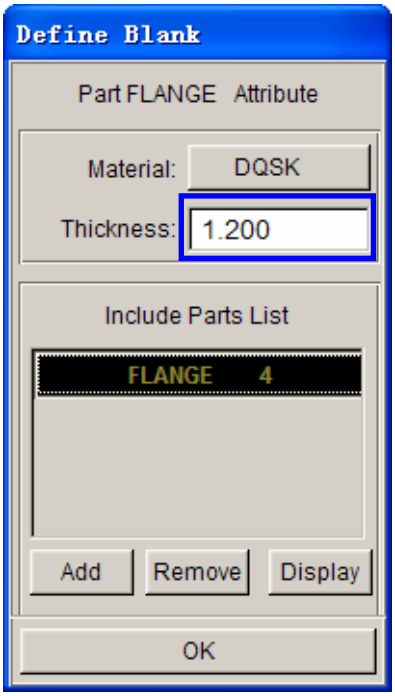


Figure 1.28: Define thickness

Note: *In MSTEP module, the simulation object is the part after forming. But defined material thickness is only the thickness of initial sheet.*

8) Click OK button in Define Blank dialogue box to return to MSTEP dialogue box. Contour line in the dialogue box that represent the sheet change from red to green. It means the sheet tool is already defined as shown figure1.29.

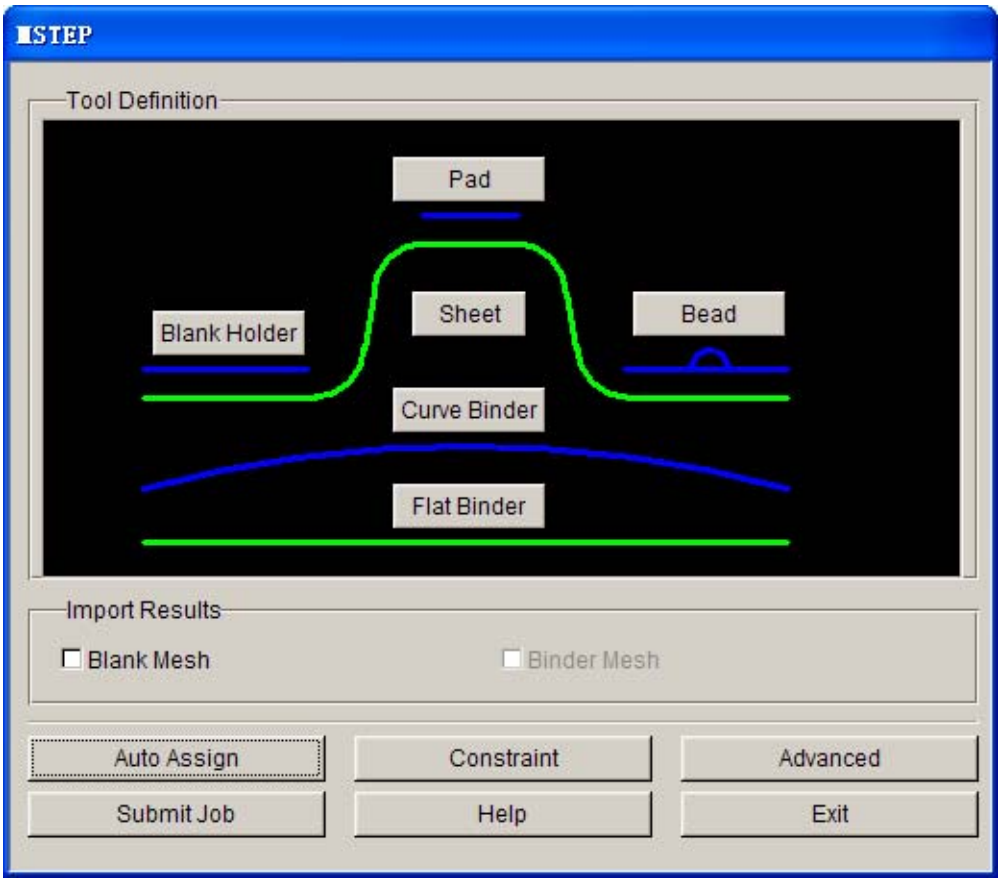


Figure 1.29: MSTEP GUI after sheet definition

9) Select Curve Binder button in MSTEP dialogue box, Define Tool dialogue box pops out. Select Add button to add corresponding part for surface (target surface).

10) Select SELECT PART mode in the popped out DEFINE TOOL dialogue box. See figure 1.30. Define binder dialogue box pops out, click Add button. See figure 1.31.

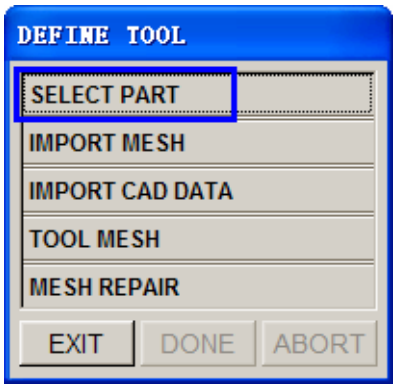


Figure 1.30: Define tool dialog box

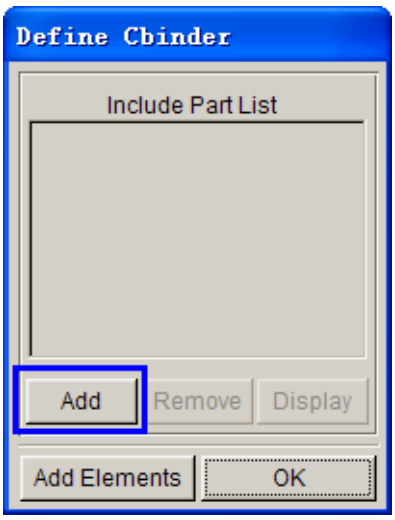


Figure 1.31: Define Cbinder dialog box

11) Select DIE2 part in the popped out Select Part dialogue box as the corresponding part of surface target. See figure 1.32.

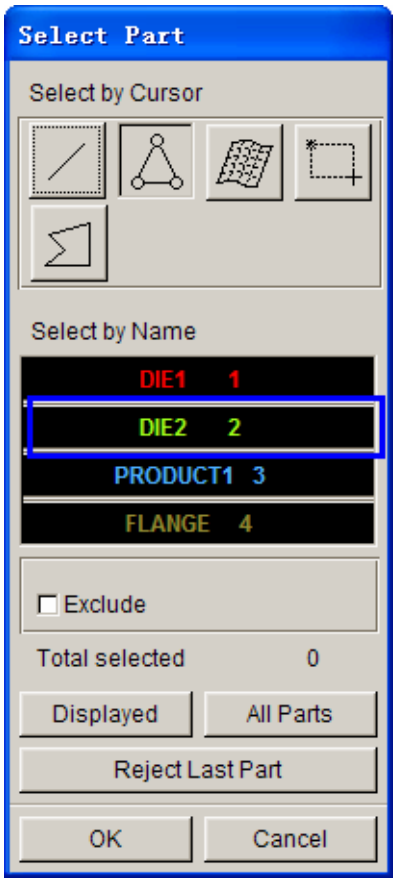


Figure 1.32: Select part dialog box

12) Click OK button, to return to Define Binder dialogue box.

13) Click OK button to return to Define Binder dialogue box. Target part selected in the last step is listed in the Include Part List. See figure 1.33.

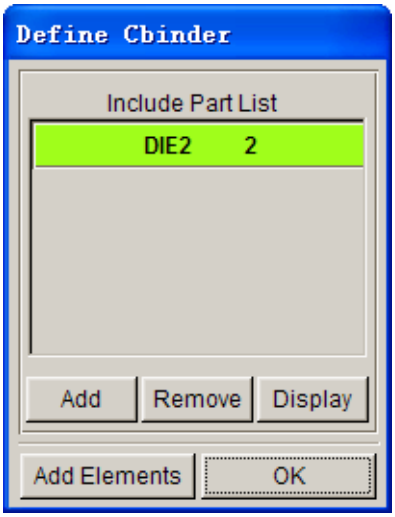


Figure 1.33: Define Cbinder dialog box

14) Click OK button to return to MSTEP dialogue box. Contour line of target surface in the dialogue box is changed from red to green, which means the tool is already defined. See figure 1.34.

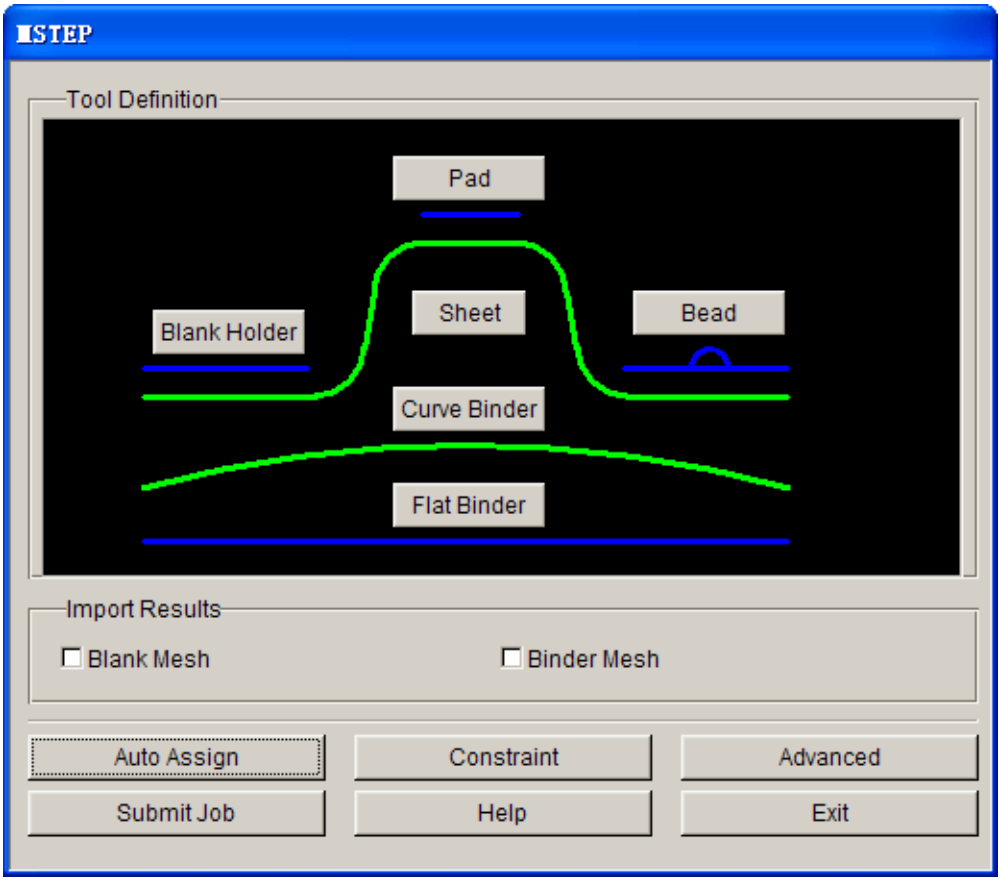


Figure 1.34: MSTEP GUI after Curve Binder definition

4. Define Boundary

Because in this example, only the flange part of the final product is opened. So user needs to defined boundary where flange and product combine. In DYNAFORM, constraints only apply on nodes. To make is more convenient to select nodes on the boundary, user may generate boundary line first and then select boundary nodes directly by selecting boundary line.

Open only FLANGE part, close other parts and set FLANGE as the current part. Select icon as the figure below.（FE Boundary Line）.See figure 1.35.

Figure 1.35: Line/Point dialog box

Click OK in the later dialogue box. Put the generated boundary line into the current part.（FLANGE）See figure1.36.

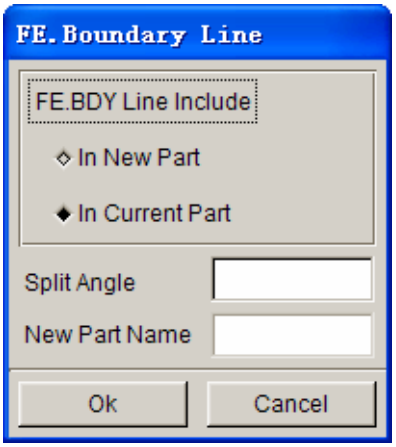


Figure 1.36: FE_Boundary Line dialog box

It is only required to constraint boundary where FLANGE and product combine to brake the consecutive boundary line. See figure 1.37.

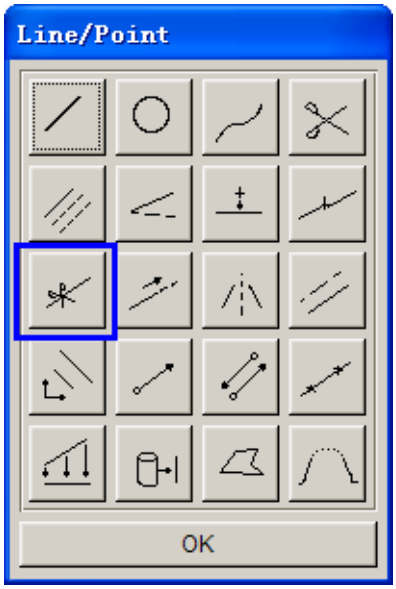


Figure 1.37: Line/Point dialog box

1) Select Constraint button in MSTEP dialogue box to define constraints. See figure 1.38.

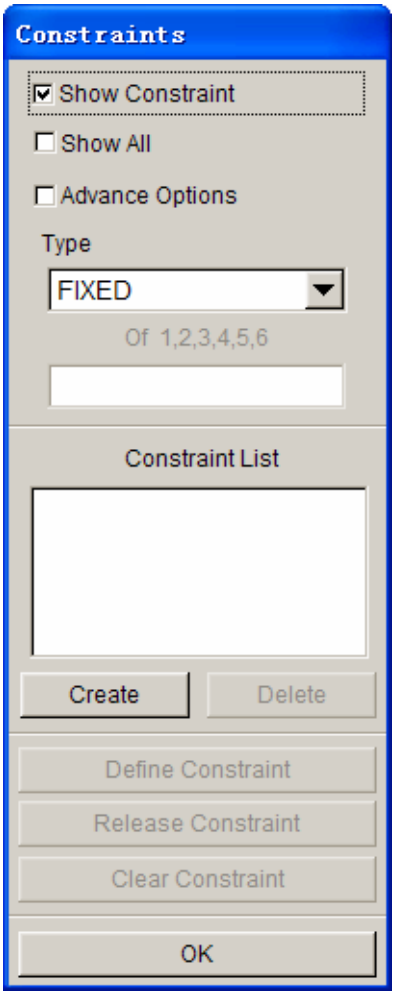


Figure 1.38: Constraints dialog box

2) In constraint Type. Default constraint is fixed constraint. Click Create button, constraint group number define dialogue box pops up. Click OK button to accept default setting. See figure 1.39.

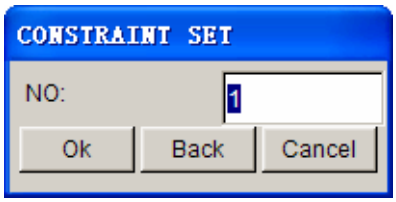


Figure 1.39: CONSTRAINT SET dialog box

3) Select New Line mode in the Select Node dialogue box. See figure 1.40.

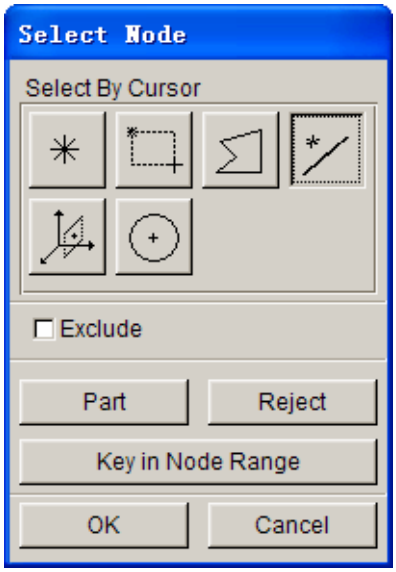


Figure 1.40: Select Node dialog box

4) Select boundary line, see figure 1.41. Check whether the selected line is correct. If it is not correct, re-break line.

Figure 1.41: Illustration of model boundary line

5) Tolerance calculation of the accept point from the line.

6) Return to Constraint dialogue box. Defined constraint group number is added into the Constraint List. And at the same time, highlight defined constraint group node member in diagram display area. See figure 1.42.

Figure 1.42: Illustration of model constraints

5. Start MSTEP Solver

Until now, all solver relevant parameters are defined in the example. Select Submit Job button in MSTEP dialogue box to start MSTEP solver for solving calculation. See figure 1.43.

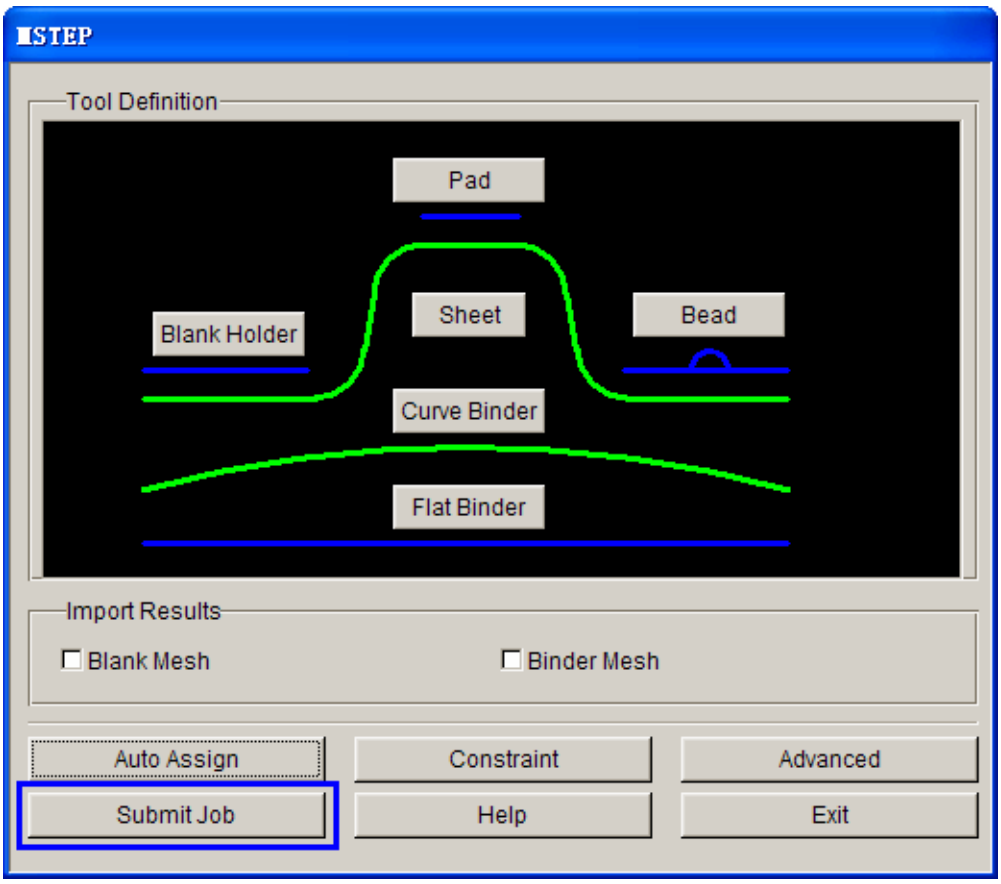


Figure 1.43: MSTEP GUI

V. Manage Calculation Result

After MSTEP calculation, DYNAFORM automatically read in blank open contour line and add it into the new part. In this example, there are two more parts: OUTLINE is the result of plane opening, and OUTLN3D is the result of surface opening.

Reserve only PRODUCT1 and OUTLN3D, close other parts. See figure 1.44.

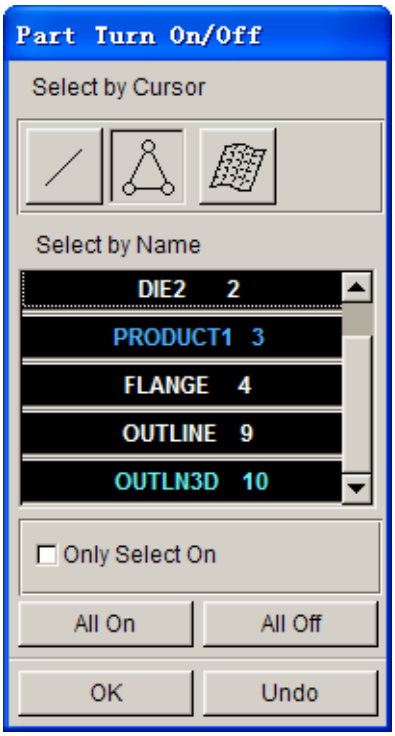


Figure 1.44: Part Turn On/Off dialog box

At the same time, close element and node. See figure 1.45.

Figure 1.45: Display options

Result refers to the figure 1.46.

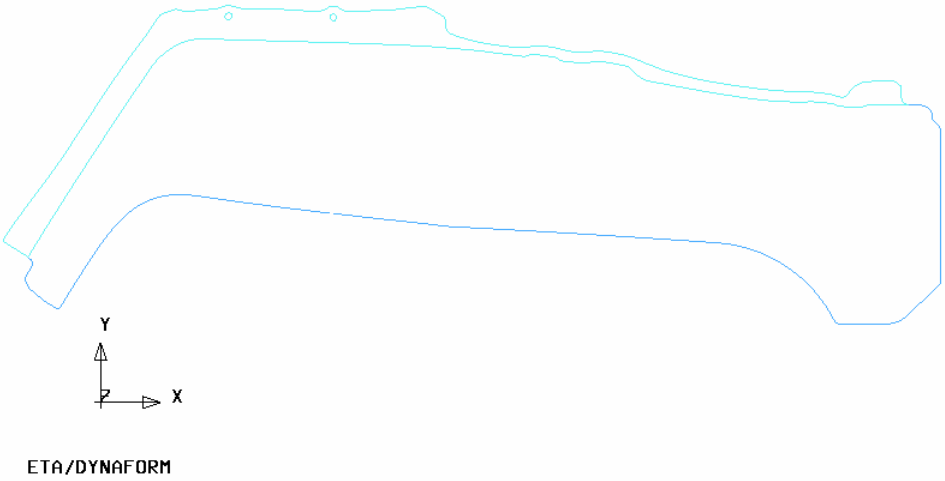


Figure 1.46: Illustration of result

Complete trim line can be created after breaking and deleting combining part in the middle, See figure1.47.

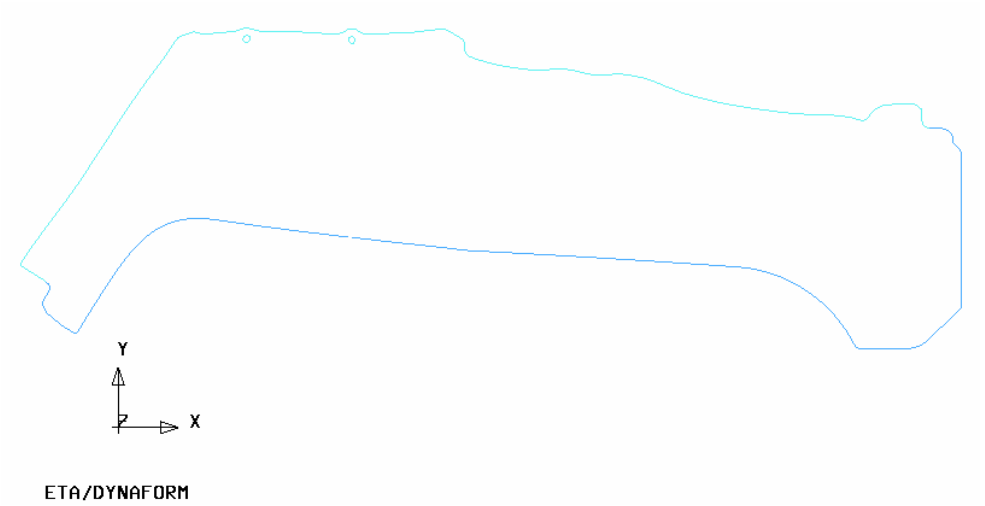


Figure 1.47: Illustration of trim line

Example 2. Blank Profile Estimation

I. 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.

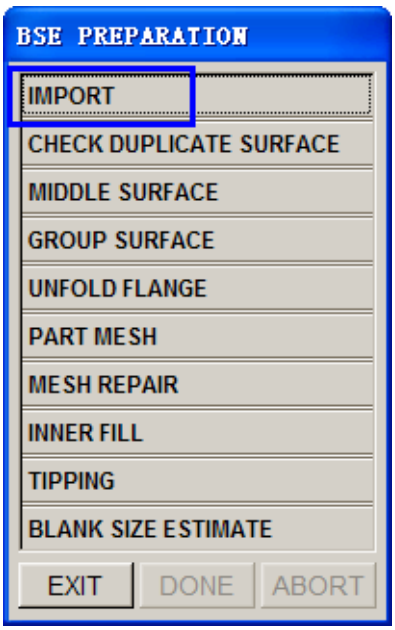


Figure 2.1: BSE preparation dialog box

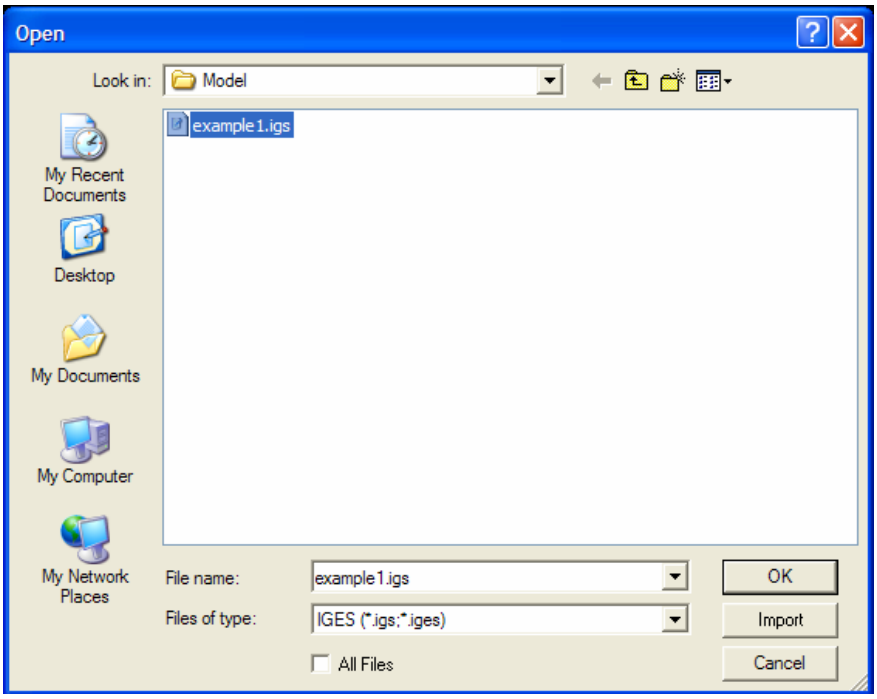


Figure 2.2: Import file window

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.

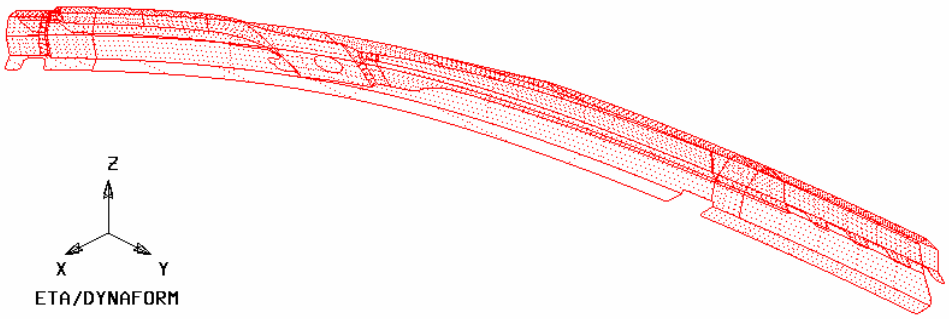


Figure 2.3: Illustration of Example 1

***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.

II. 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.

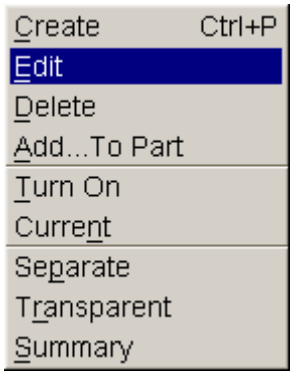


Figure 2.4: Part menu

1. Select **Parts→Edit** to display the **Edit Part** dialog box. All defied 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 compete the operation. You may also change the part color.

3. Click **OK** button to dismiss the dialog box.

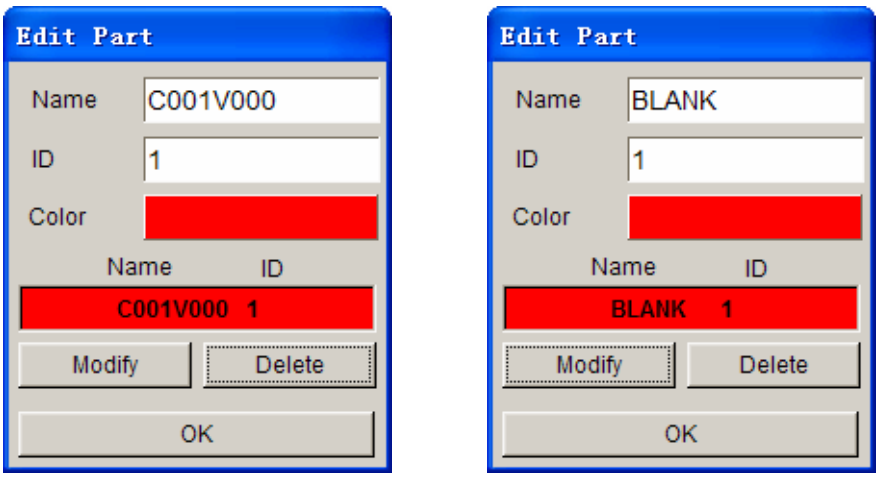


Figure 2.5: Edit part dialog box

III. 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.

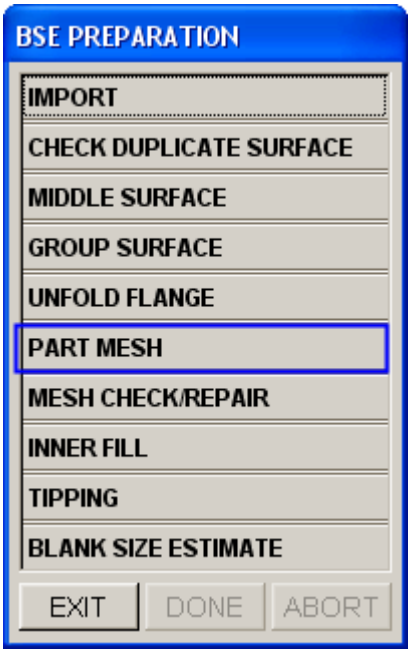
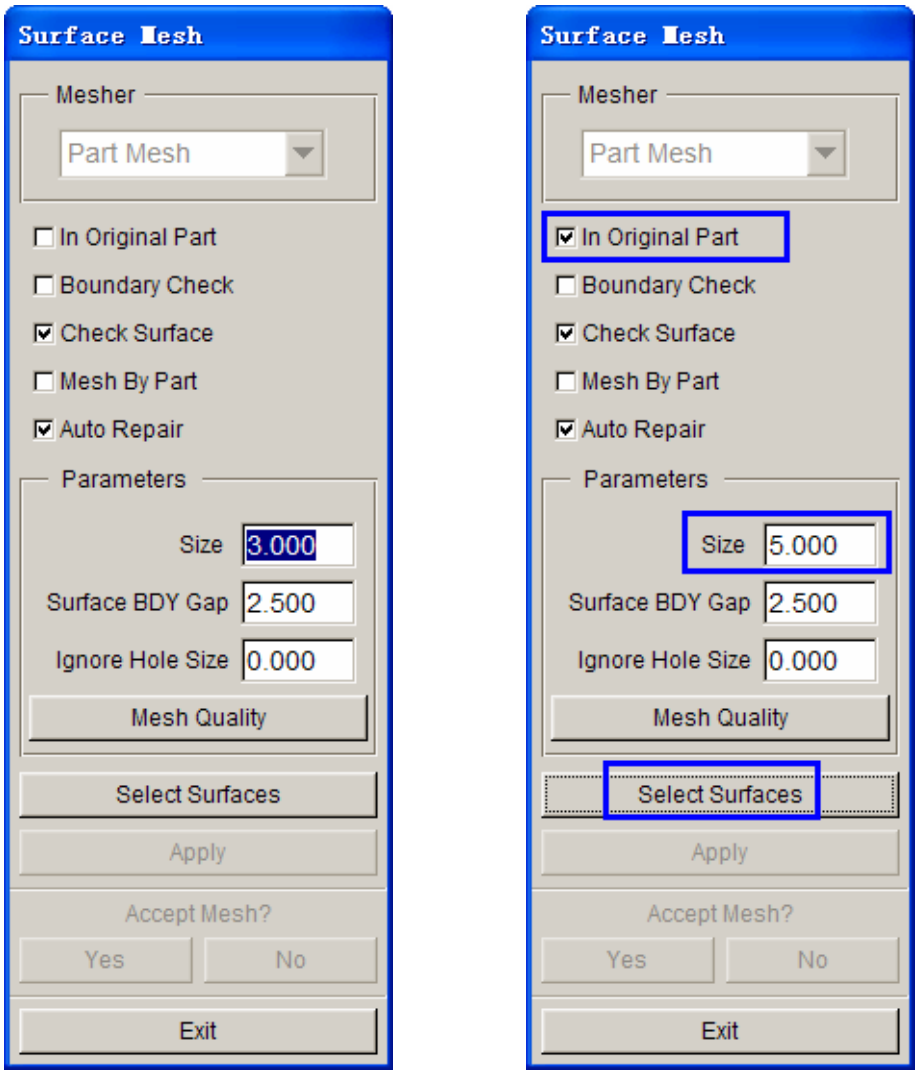


Figure 2.6: BSE preparation dialog box

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.

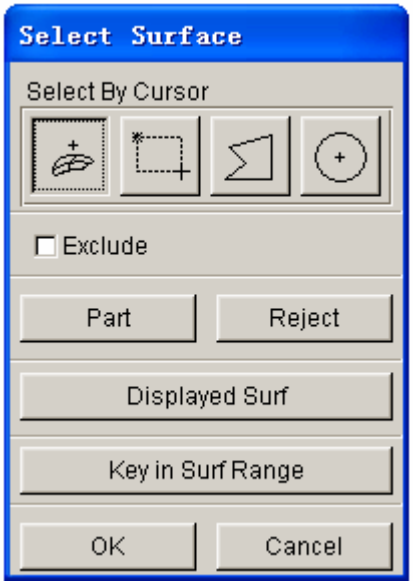


Figure 2.8: Surface mesh dialog box

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.

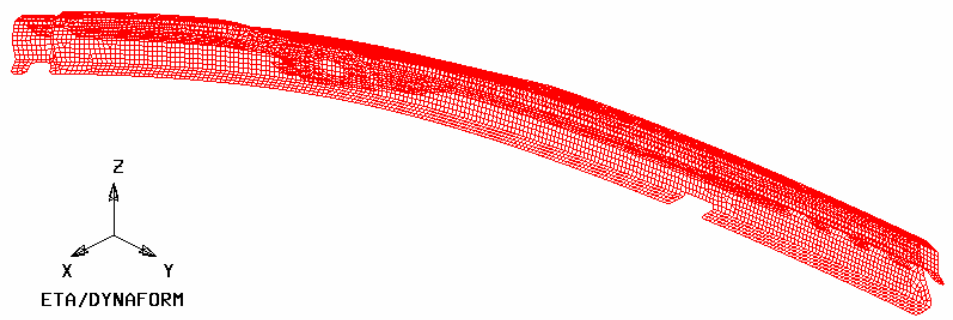


Figure 2.9: Part mesh

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.

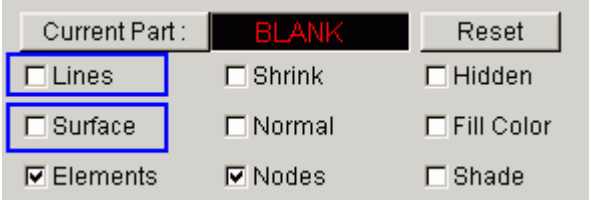


Figure 2.10: Display options

IV. 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.

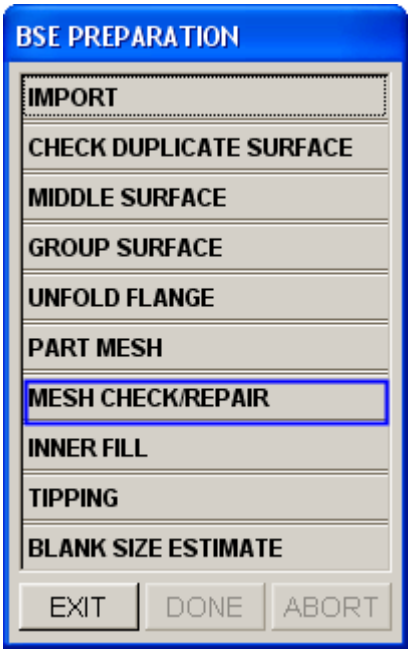


Figure 2.11: BSE preparation dialog box

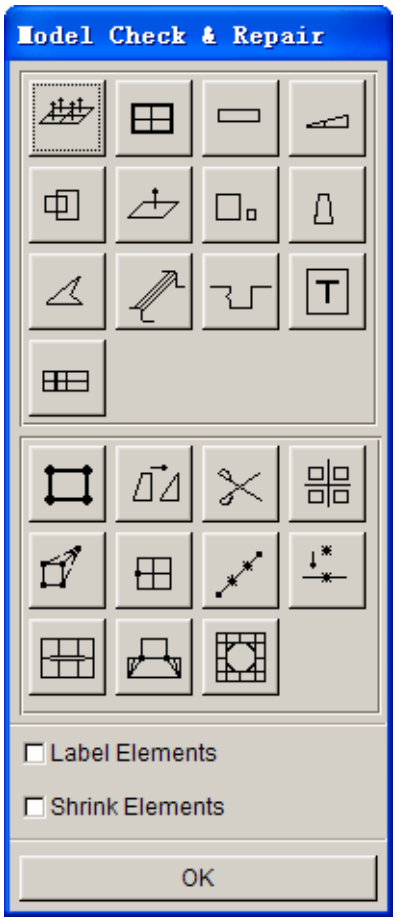


Figure 2.12: Model check & repair dialog box

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°**.

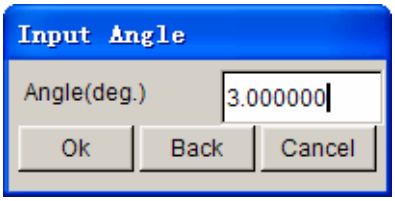


Figure 2.13: BSE preparation dialog box

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.

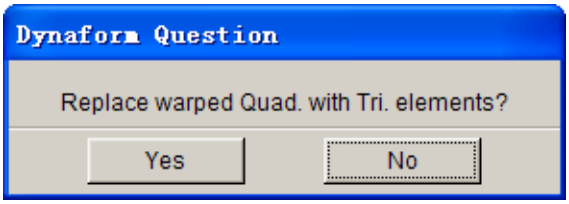


Figure 2.14: Dynaform question dialog box

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.

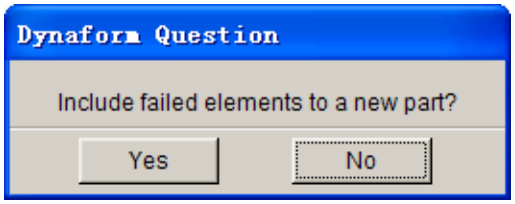


Figure 2.15: Dynaform question dialog box

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

Boundary Display 

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.

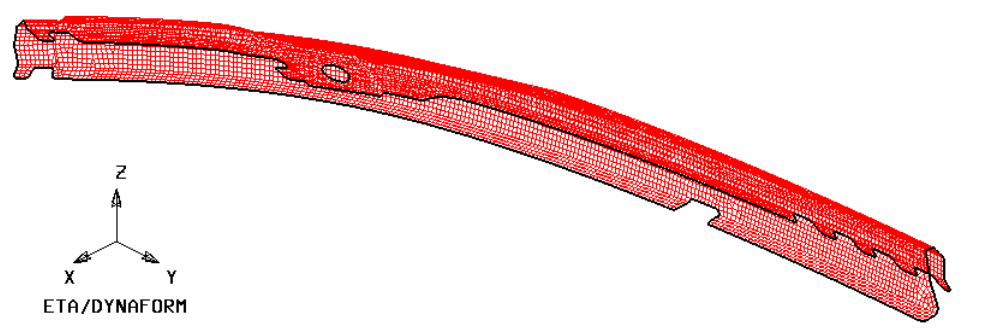


Figure 2.16: Boundary of part

2. 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.

3. 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.

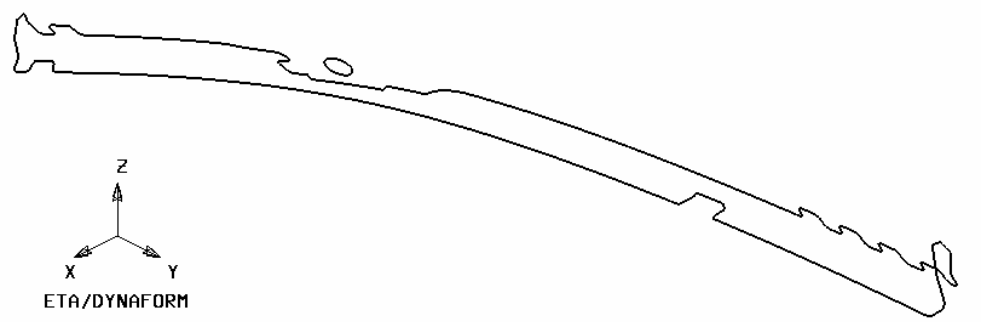


Figure 2.17: Dynaform question dialog box

4. Use other checking functions, check and delete element with too small size and overlap elements.
5. Click on  icon in the **Icon bar** to refresh the screen.
6. 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.

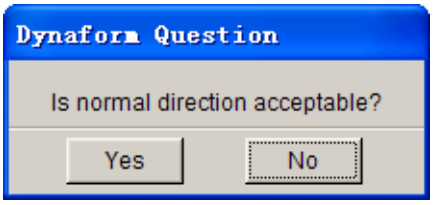


Figure 2.18: Dynaform question dialog box

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.

V. MSTEP Module and Parameter Setup

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

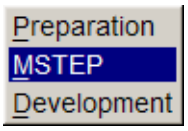


Figure 2.19: BSE menu

2. The MSTEP GUI is illustrated in Figure 2.20.

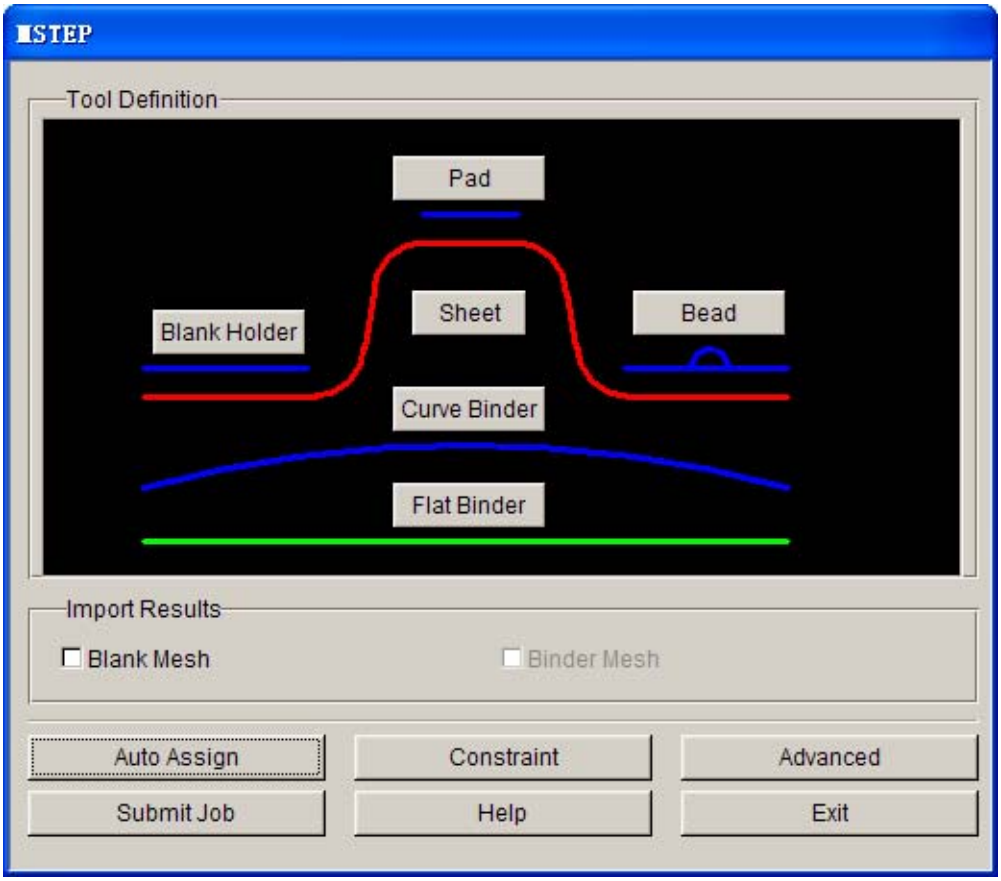


Figure 2.20: MSTEP GUI

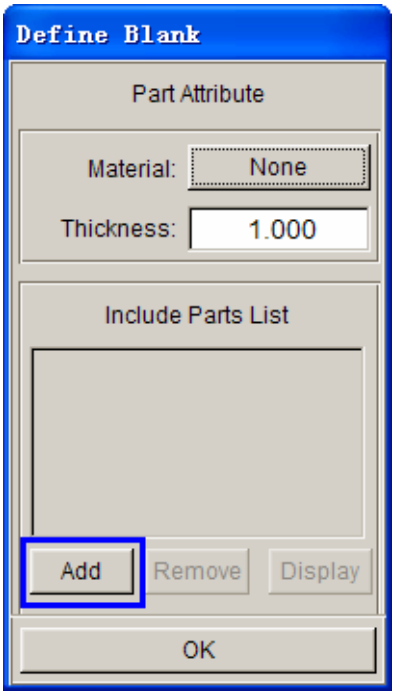


Figure 2.21: Define blank dialog box

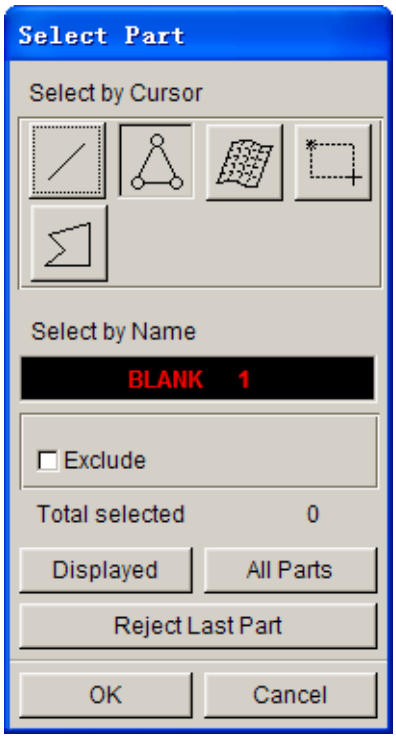


Figure 2.22: Select part dialog box

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.

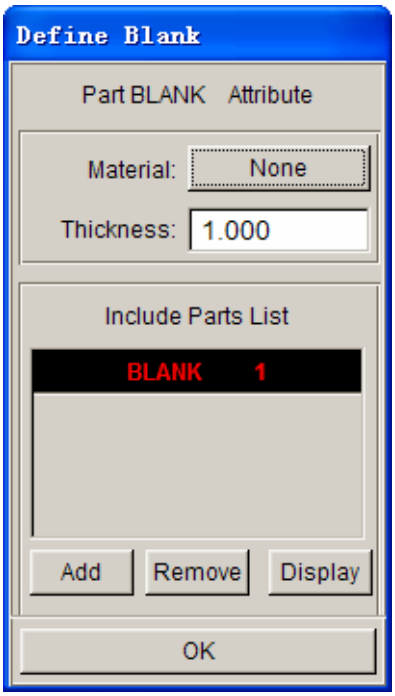


Figure 2.23: Define blank dialog box

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.

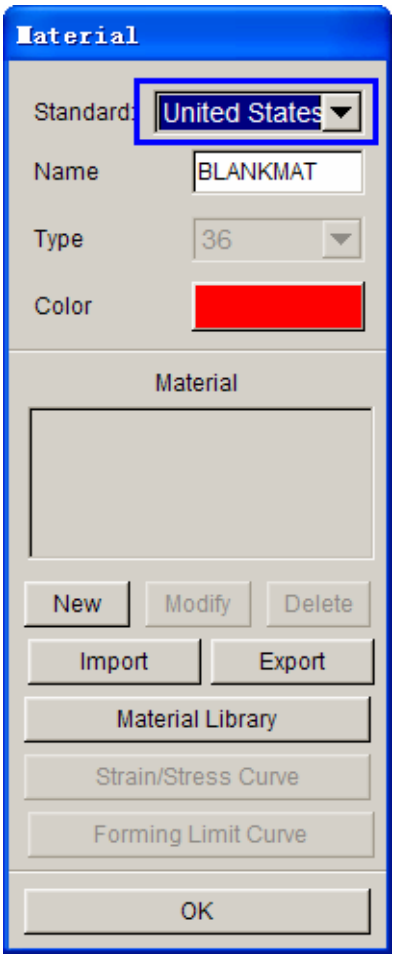


Figure 2.24: Material dialog box

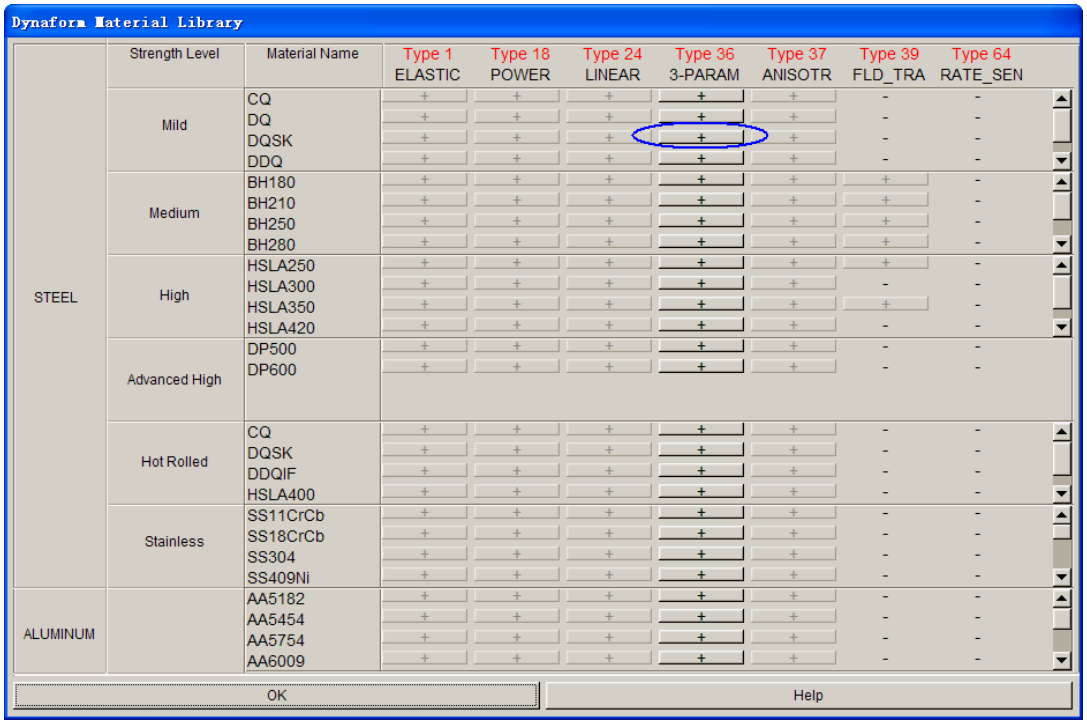


Figure 2.25: Material library window

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).

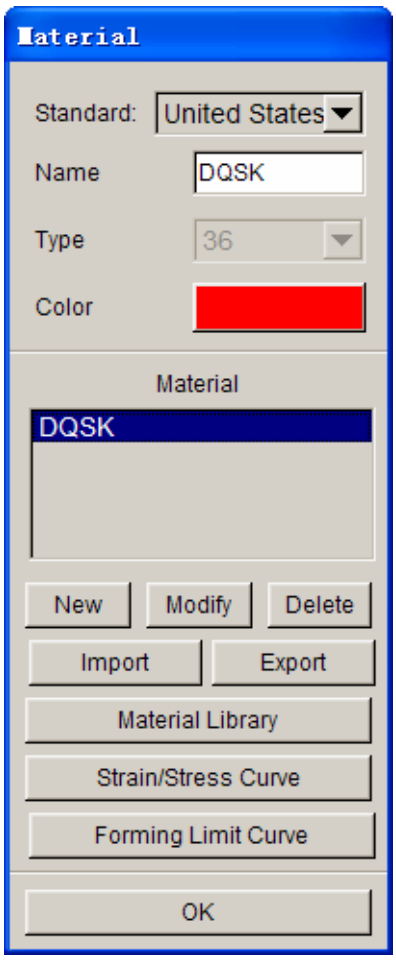


Figure 2.26: Material dialog box

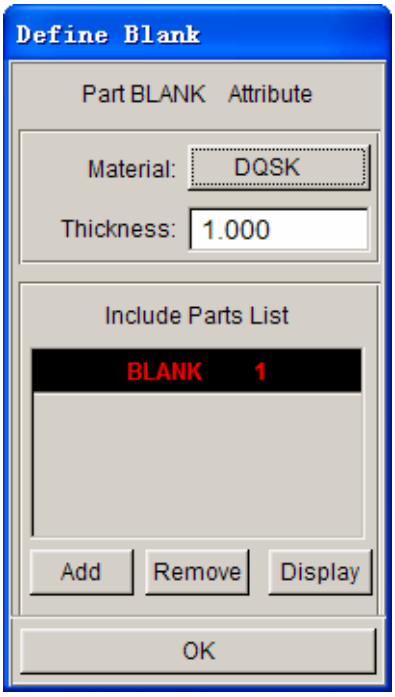


Figure 2.27: Define blank dialog box

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.

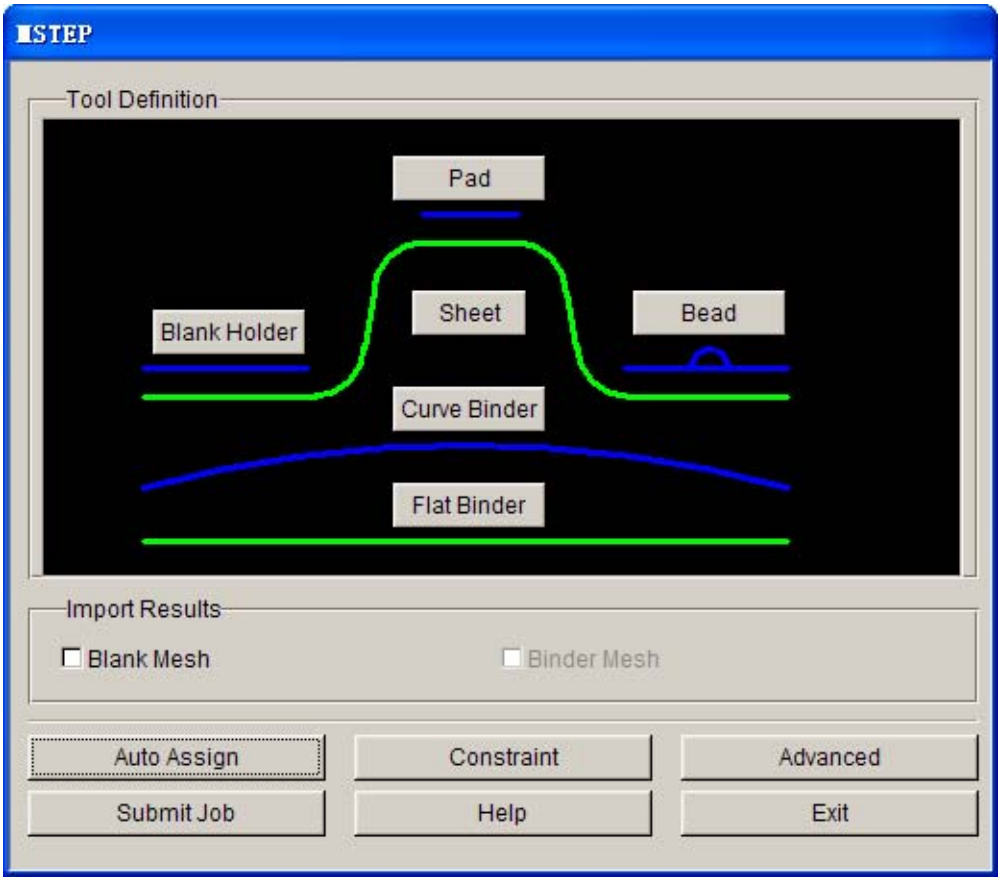


Figure 2.28: MSTEP GUI after sheet definition

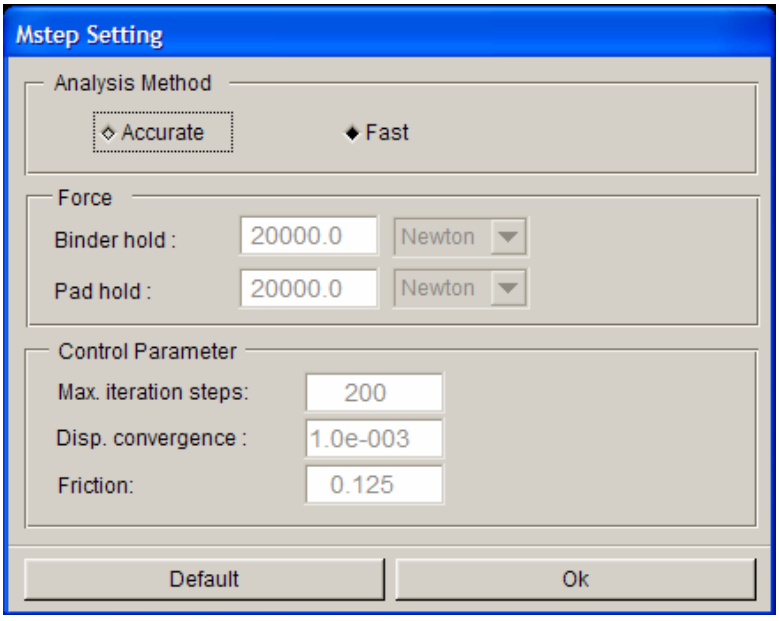


Figure 2.29: MSTEP Setting dialog box

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.

VI. 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.

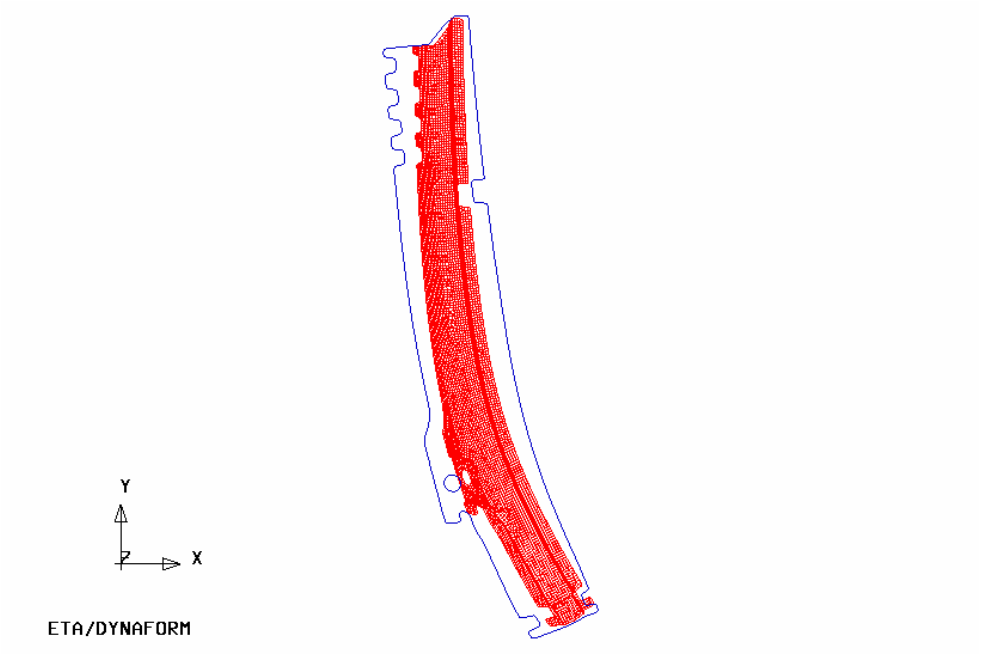


Figure 2.30: Blank outline

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

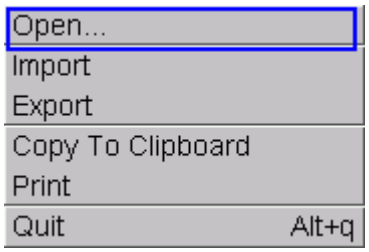


Figure 2.31: File manager

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

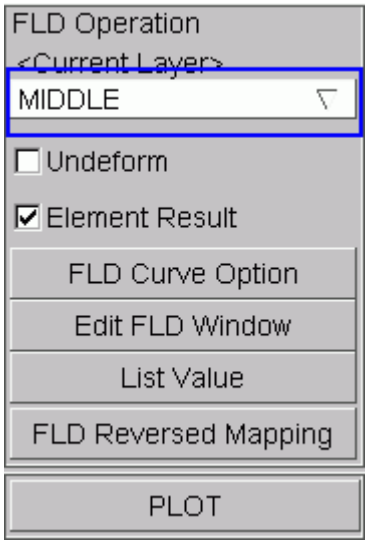


Figure 2.34: FLD dialog

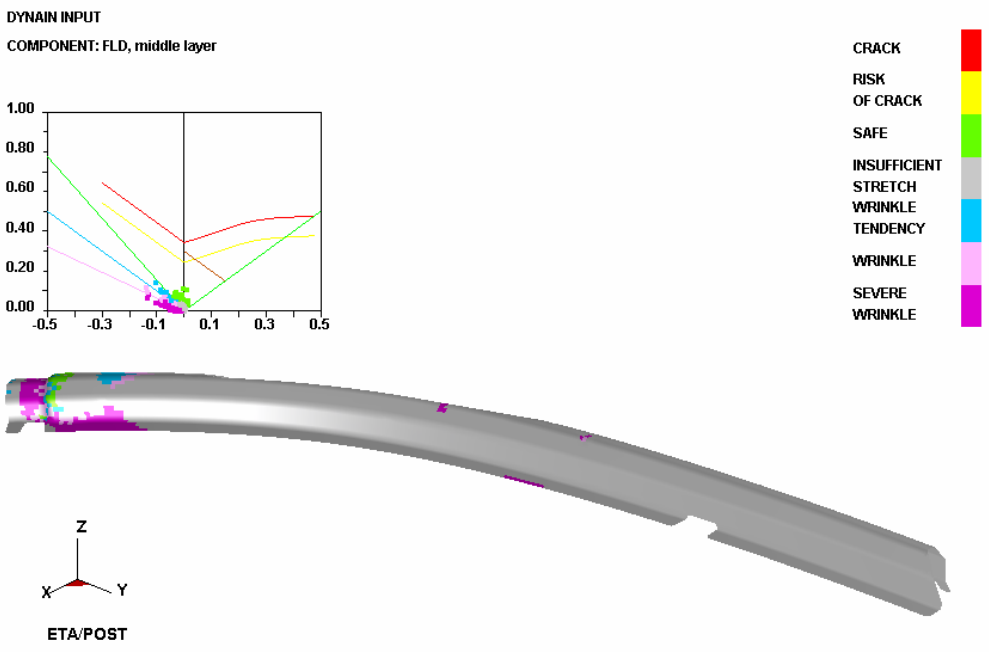


Figure 2.35: FLD distribution

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

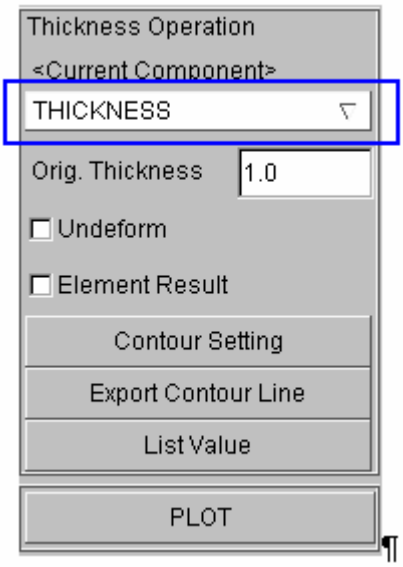


Figure 2.37: Thickness operation dialog

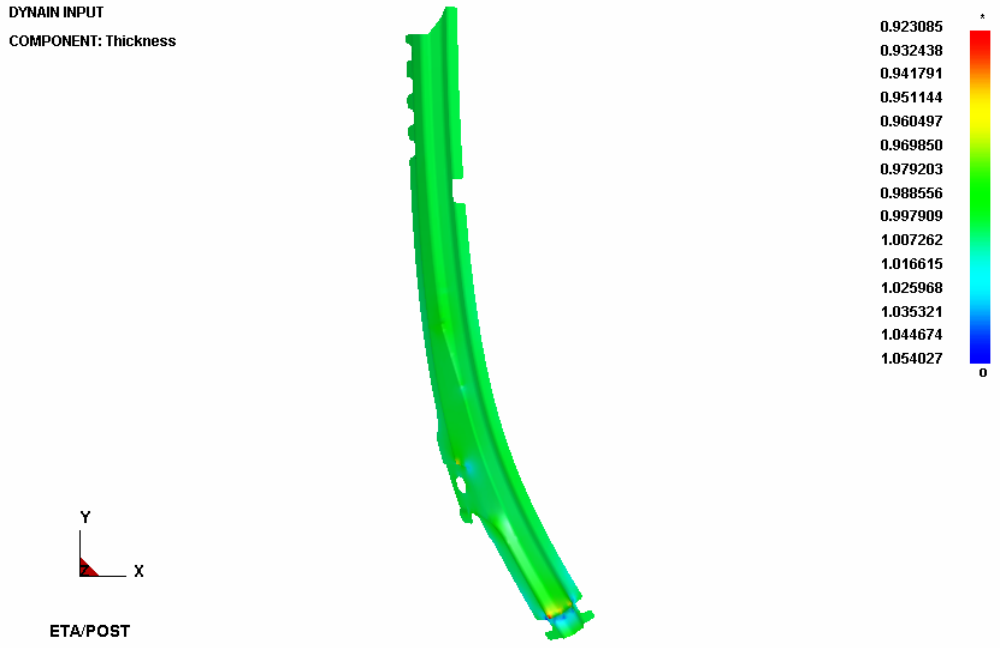


Figure 2.38: Thickness/thinning contour display

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.

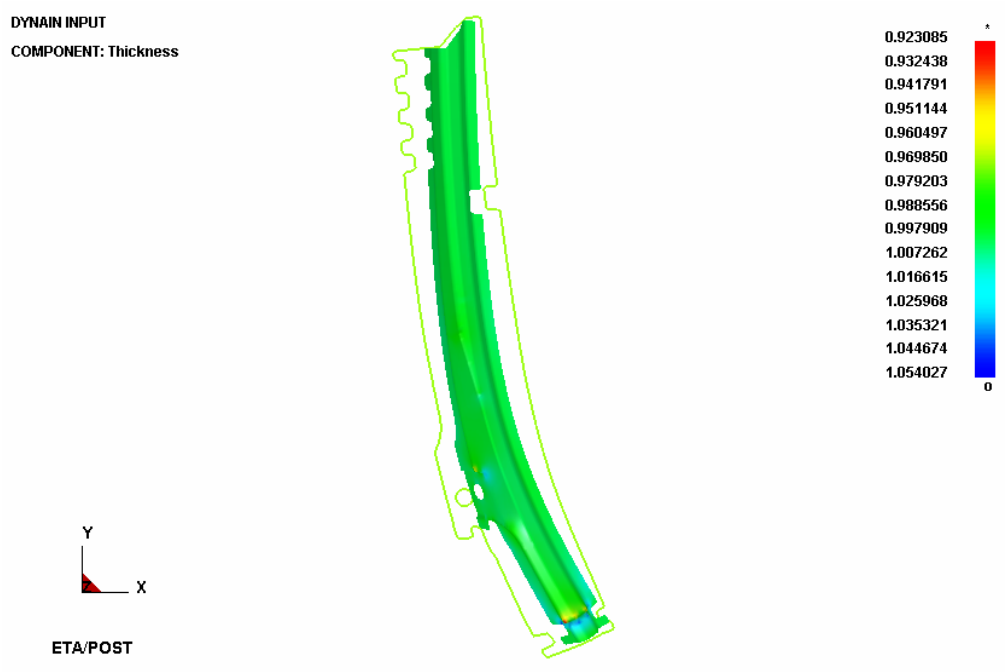


Figure 2.39: Blank outline

VII. 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.

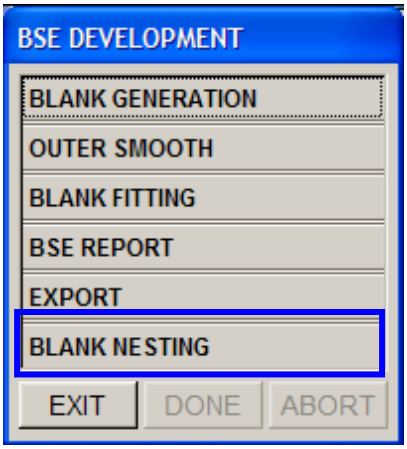


Figure 2.40: BSE development dialog box

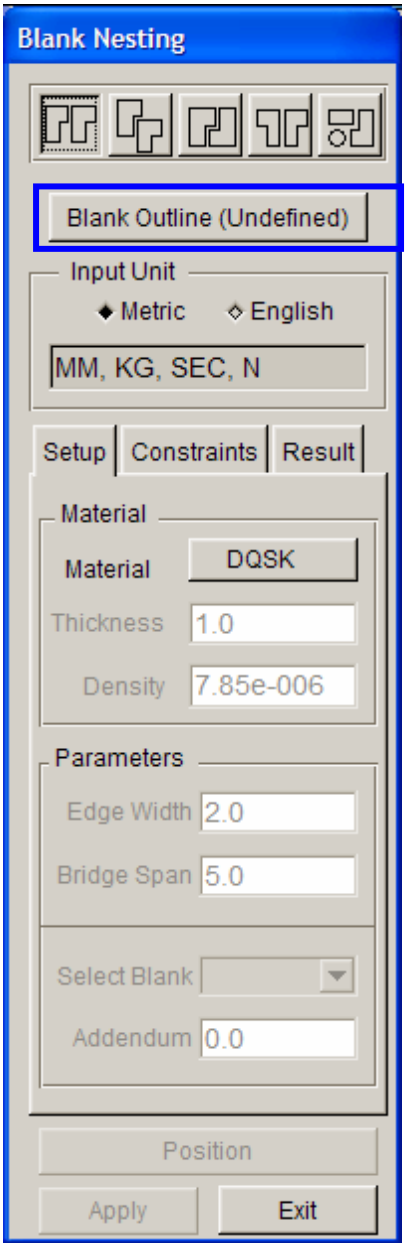


Figure 2.41: Blank nesting dialog box

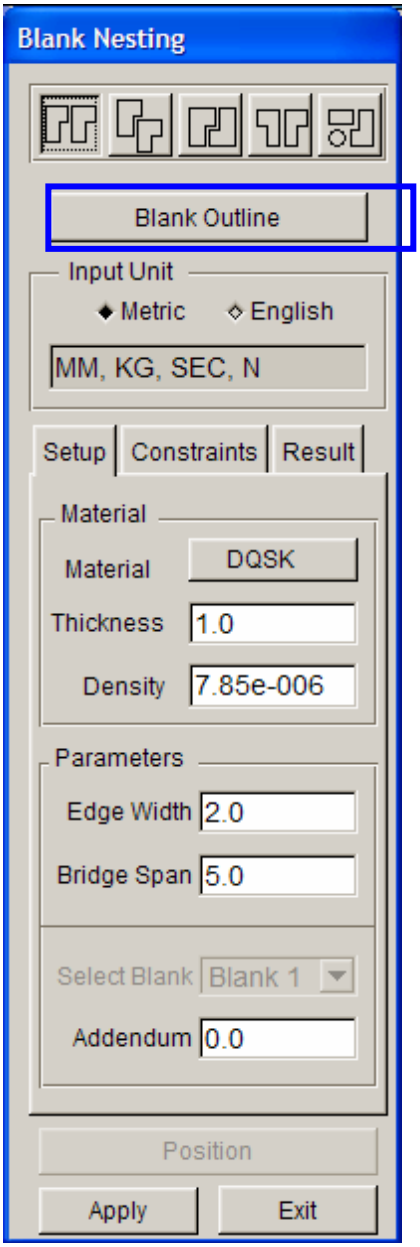


Figure 2.42: Blank nesting dialog box after line selection

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.

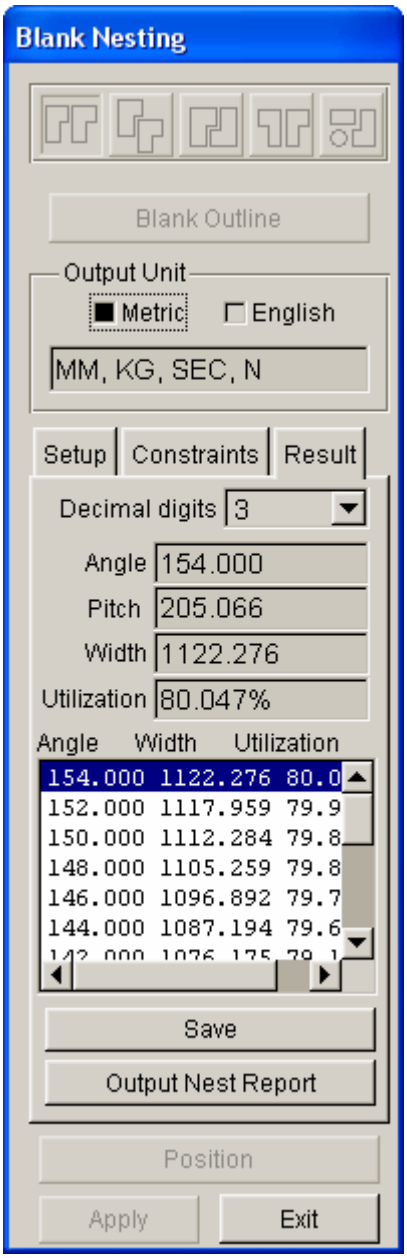


Figure 2.43: Result page

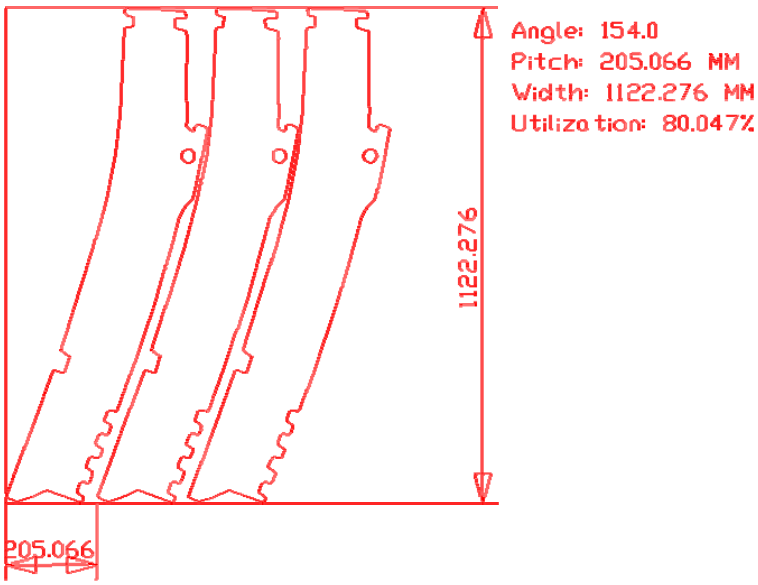


Figure 2.44: Nesting layout

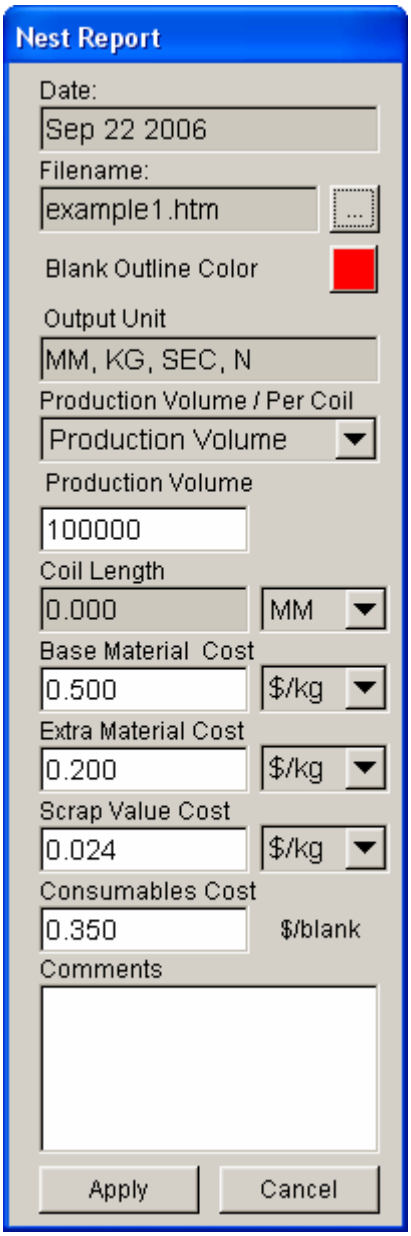


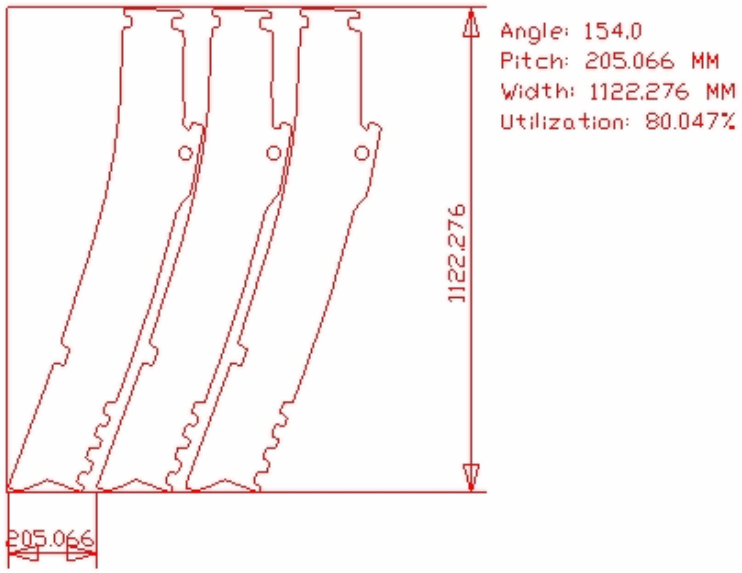
Figure 2.45: Nest report dialog box

Nesting Estimation Report

Date:	Sep 22 2006	
File Name:	example1.htm	
File Units:	MM, KG, SEC, N	
Material Type	DQSK	
Material Thickness	1.0	mm
Bridge Span	5.0	mm
Edge Width	2.0	mm
Addendum	0.0	mm
Production Volume	100000	
Coil Length & Weight	0.0 mm & 0.0	kg
Base Material Cost	0.5	$/kg
Extra Material Cost	0.2	$/kg
Scrap Value Cost	0.024	$/kg
Consumables Cost	0.35	$/blank

Additional Comments:

Nesting Layout:



RESULTS: Single Blank, One Up Orientation

Pitch	205.066	mm
Width	1122.276	mm
Material Utilization	80.047	%
Fall Off	19.953	%
Rotation Angle	154.0	deg
Net Weight / Blank	1.446	kg
Gross Weight / Blank	1.807	kg
Fall-Off Weight / Blank	0.36	kg
Blank Perimeter	2995.261	mm
Minimum Blank Force	89.858	ton
Number of Coils	0(0.0)	
Total Material / Blank	1.606	$ / blank

Total Cost Of Production	160597.401	$
Total Scrap Value	865.144	$

Report generated by ETA/DYNAFORM-BSE, www.eta.com

Figure 2.46: Nesting report

Example 3. Formability Analysis

I. Create DataBase and Read in Model File

Start up eta/DYNAFORM 5.5

Workstation and Linux user may enter the command of “df552”（default）in the command line to start up DYNAFORM5.5. PC user may double click DYNAFORM 5.5（DF55）icon or select DYNAFORM from the program menu to start up it.

Once starting up eta/DYNAFORM, the program automatically creates the empty database file of Untitled.df in default. It’s necessary for user to import CAD or CAE model to start working.

1. Select menu **File→Import** as shown figure 3.1or click the IMPORT icon 

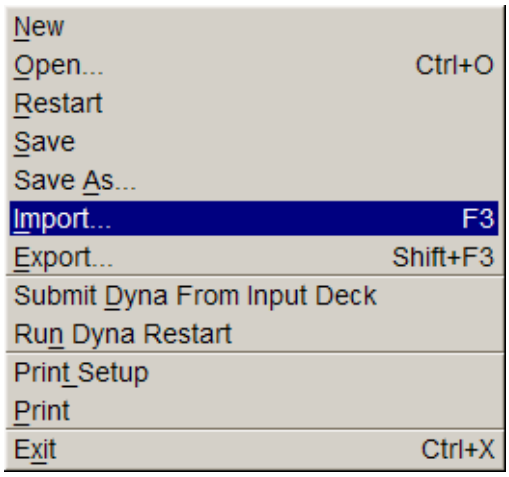


Figure 3.1: Import file menu

Find the directory of training file, and import the file of MSTEP_model2.dat into the database. See figure 3.2.

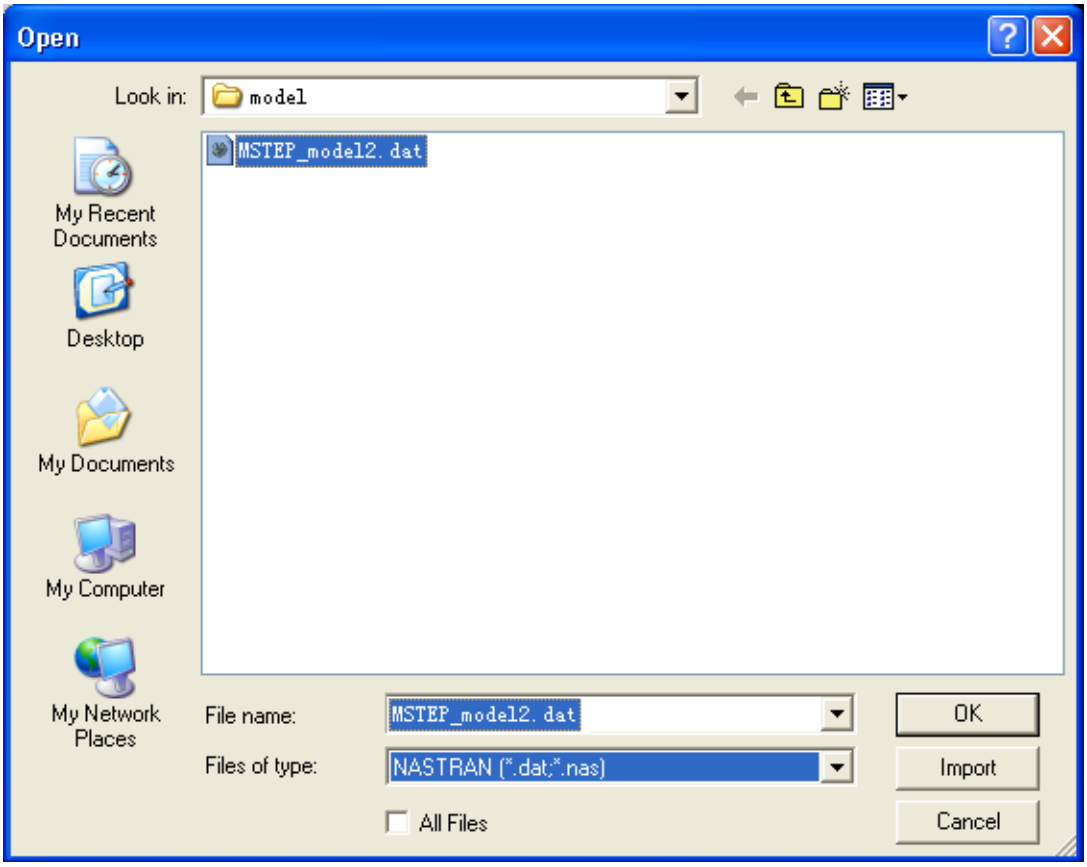


Figure 3.2: Import file dialog box

After importing the file, check the displayed picture to make sure it consist with the model shown as figure 3.3. In the example, the input model has been meshed, which is shown as a top view in the screen.

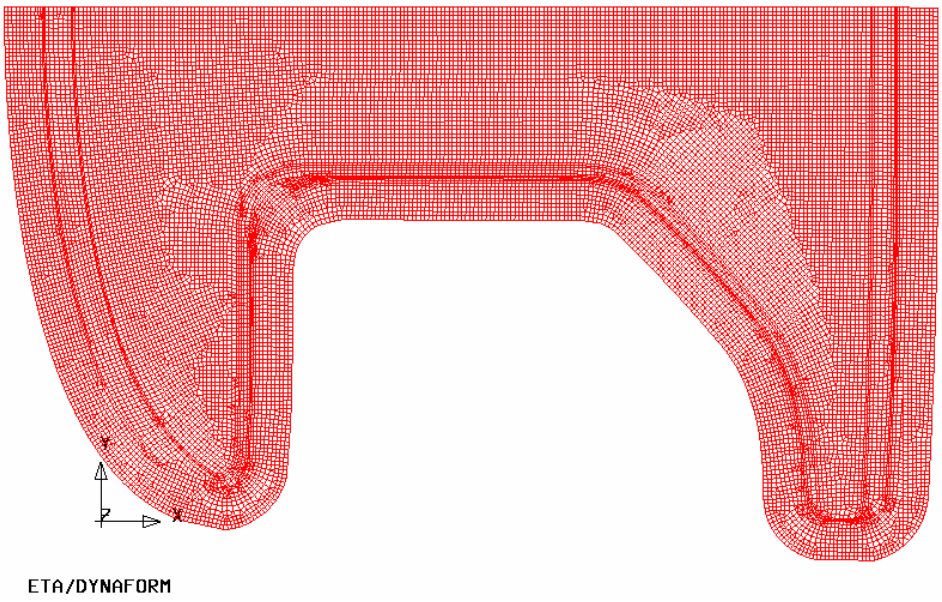


Figure 3.3: Illustration of MSTEP_model2

Note: *the icon is different in the different system platform; the other icons in the toolbar will be discussed*

in the following. User may refer to the eta/DYNAFORM Manual to obtain all the functions of the toolbar.

2. Save the database file in a specified working directory. Select menu **File→Save as** or click the SAVE icon  , input ”MSTEP_model2.df”, then select “save” to save it and exit the dialogue window. See figure 3.4.

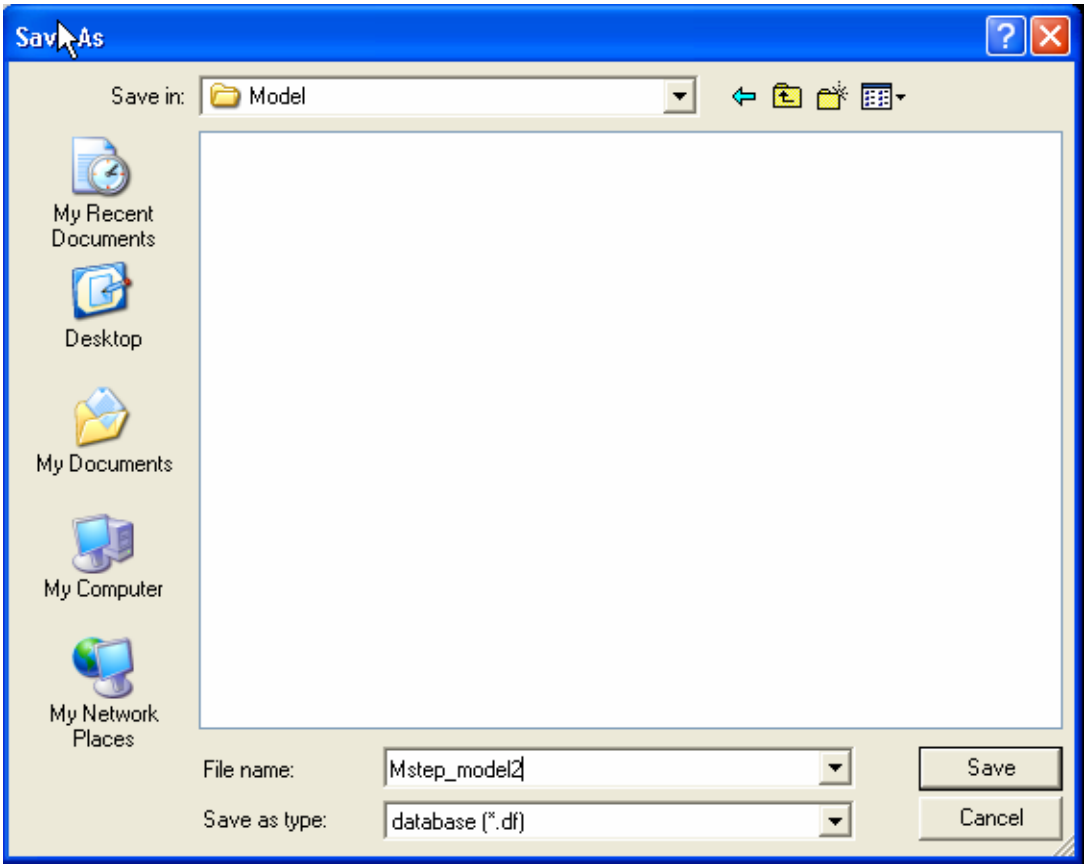


Figure 3.4: Save as dialog box

User may refer to the eta/DYNAFORM Manual to obtain the detail information about the units and file type of DYNAFORM database.

II. Edit Parts in Database

In eta/DYNAFORM, all the models are managed based on the part. All the entities will be created or read in the part in default. User may refer to the Eta/DYNAFORM Manual to obtain the detail information about the operation of the part.

The command of Edit Part is used for editing the part’s property or deleting the part. See figure3.5.

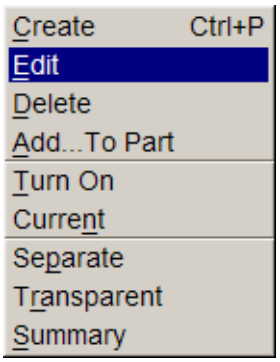
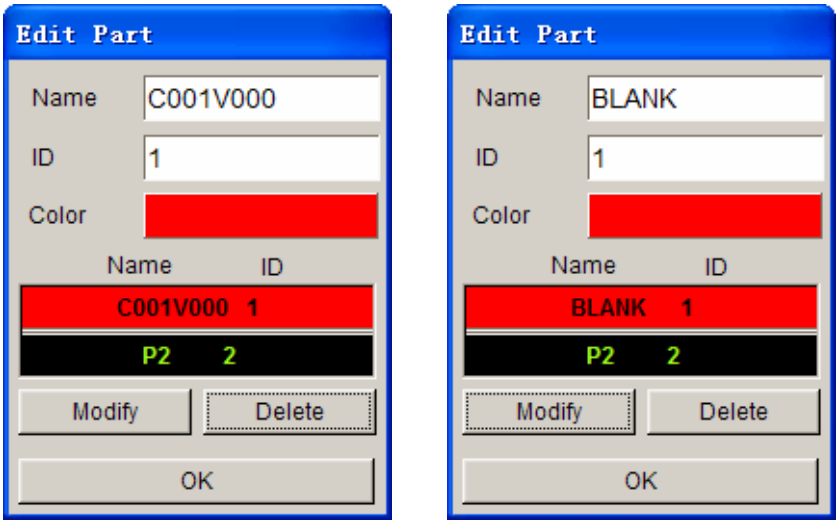


Figure 3.5: Edit menu

4. select menu **Parts→Edit**, then pop up the Edit Part dialogue window. All the defined parts are displayed in the list and indicated by their name and ID. User can change the name and ID, and delete some parts form the database.

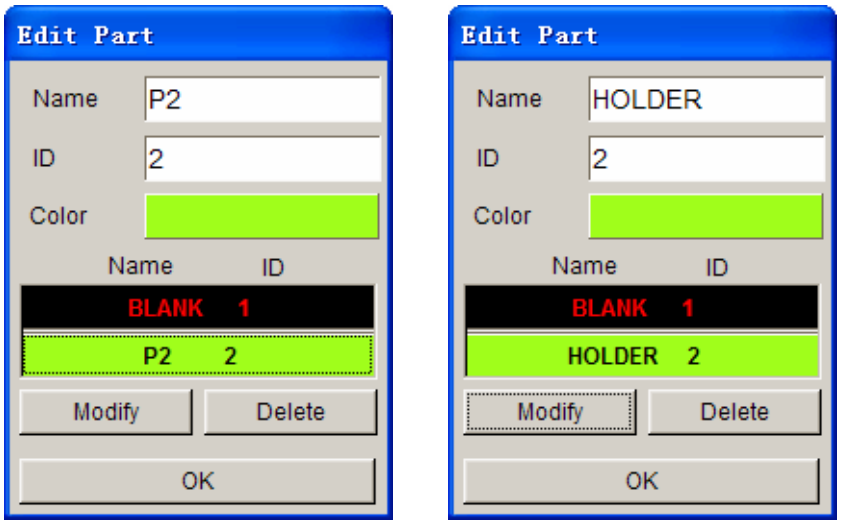
5. Select C001V000 from the part list. As shown in the following figures, user can input “BLANK” in the input box behind Name and don’t change the part color, then click the Modify button in the down-left corner to affirm these edit. See figure 3.6.



a) Before edit b) After edit

Figure 3.6: Edit Part dialog box

6. Using the same operation mode, change the part’s name of P2 into HOLDER, the result is as figure3.7.



a) Before edit b) After edit

Figure 3.7: Edit Part dialog box

7. Click “Close” to exit the edit operation.

III. Mesh Check

In this example, the input model is a mesh model in Nastran format, so user need not mesh. But it’s necessary to check the mesh quality, in order to prevent that the meshes have some potential defects impacting the simulation. All the tools used to check the mesh quality lie in the menu of **Preprocess→Model Check. / Repair** User may check the meshes by selecting menu **Preprocess→Model Check / Repair** or using the shortcut key **Ctrl + R**. See figure 3.8

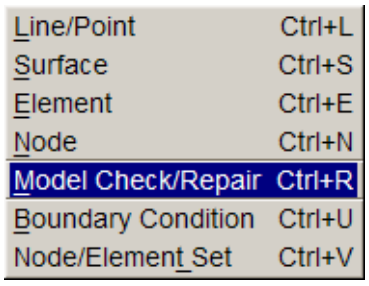


Figure 3.8: Model Check/Repair menu

Open the Model Check dialogue window as shown in the following figure 3.9.

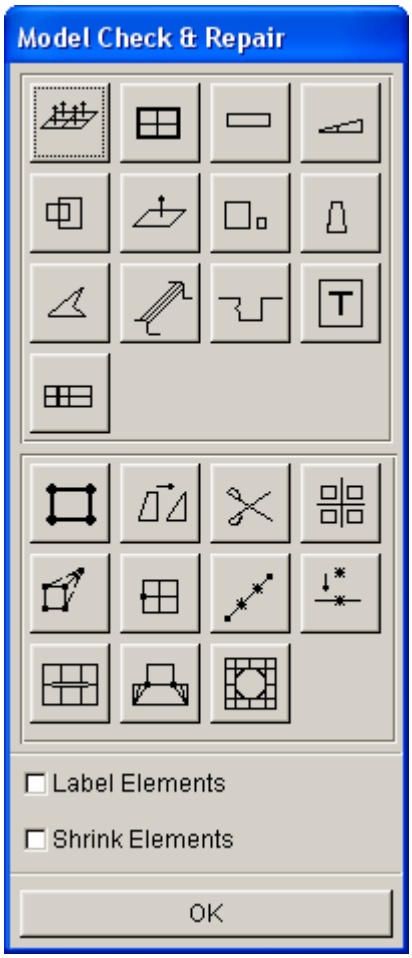


Figure 3.9: Model Check/Repair dialog box

As shown above, the Model Check dialogue window comprises some functions helping user to check the mesh quality. In this example, the functions used are mainly element interior angle check, element check warpage, model boundary display, and auto plate normal. Other functions can be learned from the eta/DNAFORM manual.

The check standards of mesh model are as follow:

- Element interior angle > 5.0 degrees, the failed elements will be deleted or merged.
- Element warp angle < 3.0 degrees， the failed elements will be divided into triangle elements by system.
- The model boundary should be a closed curve; the mesh model can’t have gaps and holes, as well as the degradation elements.
- The normal direction should be adjusted consistent for all elements.

In this example, the input mesh model should accord with the check standard, and shouldn’t exist the failed elements.

IV. MSTEP Module and its Parameter Set

1. select menu **BSE→MSTEP** to open the MSTEP solution module. See figure 3.10.

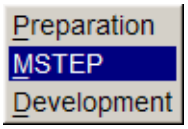


Figure 3.10: BSE menu

2. The following figure 3.11 is the Schematic View of MSTEP module. The module operation is simple, once assign the corresponding part to tool and select the solution mode; the simulation operation can be performed.

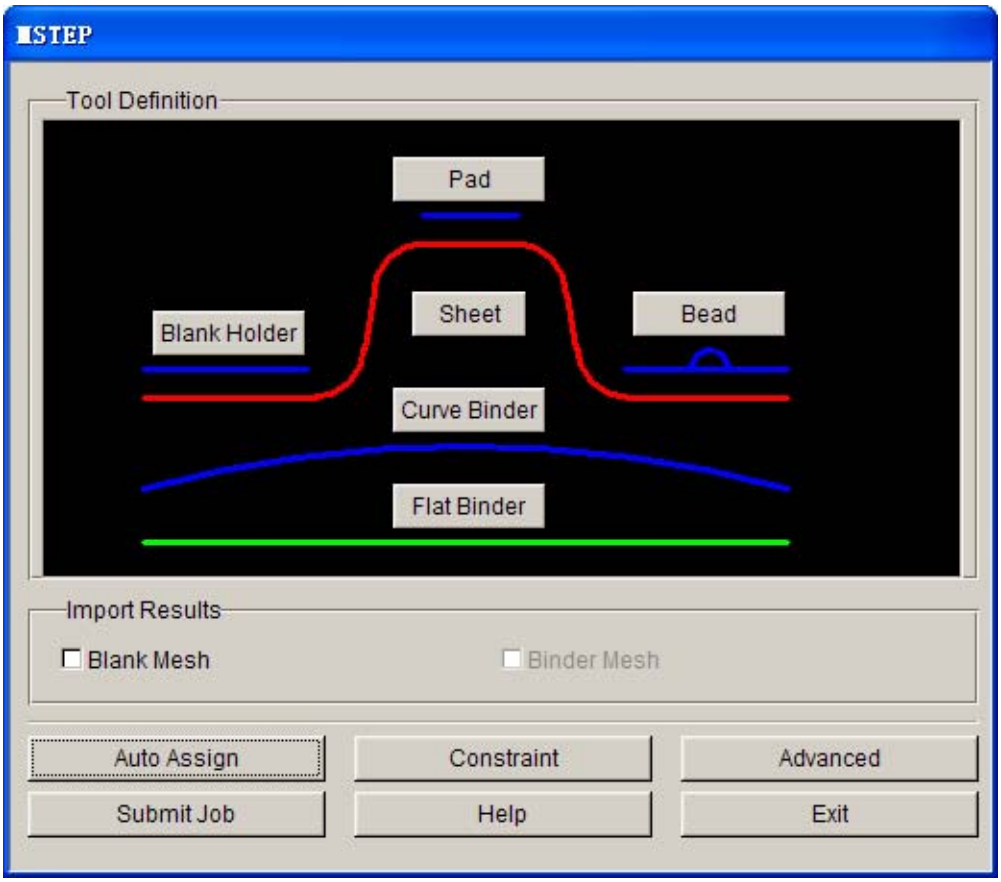


Figure 3.11: MSTEP GUI

3. define tool

   1) Select “Sheet” from the MSTEP dialogue window, then pop up the Define Blanks dialogue window; select “Add” to add the corresponding part to the sheet-metal tool. See figure 3.12.

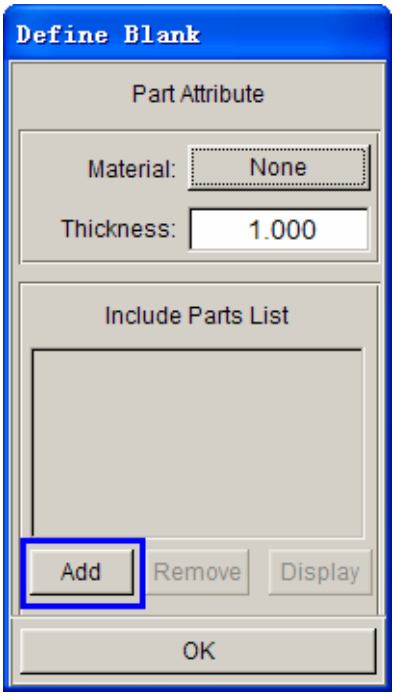


Figure 3.12: Define Blank dialog box

2) In the pop-up Select Part dialogue window, select the “BLANK” as the part corresponding to the sheet-metal tool, as shown in following figure 3.13.

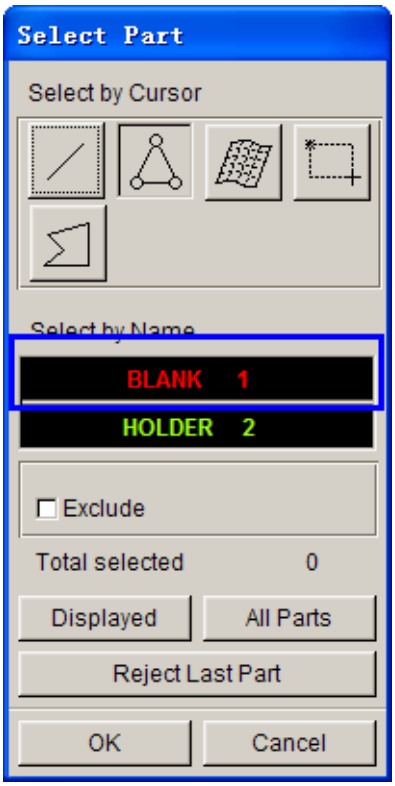


Figure 3.13: Select Part dialog box

3) Click “OK” to return to the Define Blank dialogue window, user can find the selected target part has been added to the Include Parts List. See figure 3.14.

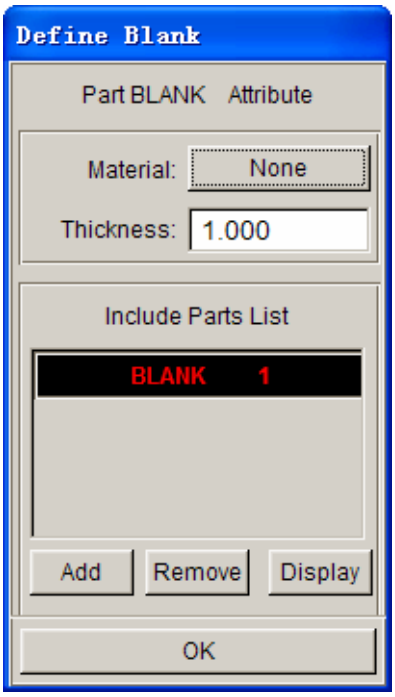
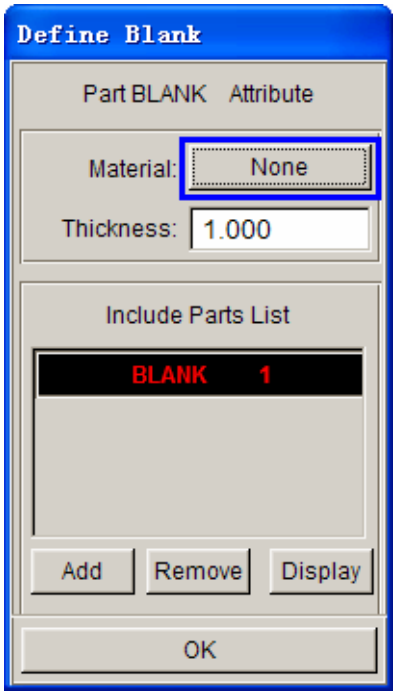


Figure 3.14: Define Blank dialog box

4) Click the BLANK from the Include Parts List, the selected part’s color is changed into black automatically; then select the button of None behind Material to pop up the Material dialogue window, at the time all the elements in the selected part are highlighted. See figure 3.15~3.16.



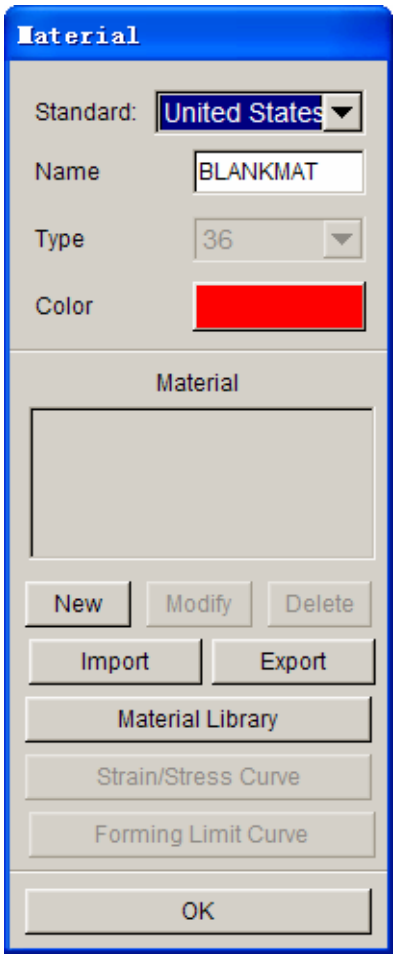


Figure 3.15: Define Blank dialog box

Figure 3.16: Define Material dialog box

5) Select the Material Library button from the Material dialogue window, then pop up the dialogue window of Dynaform Material Library, from which select the material of DQSK type 36 as the material type of BLANK part. See figure 3.17.

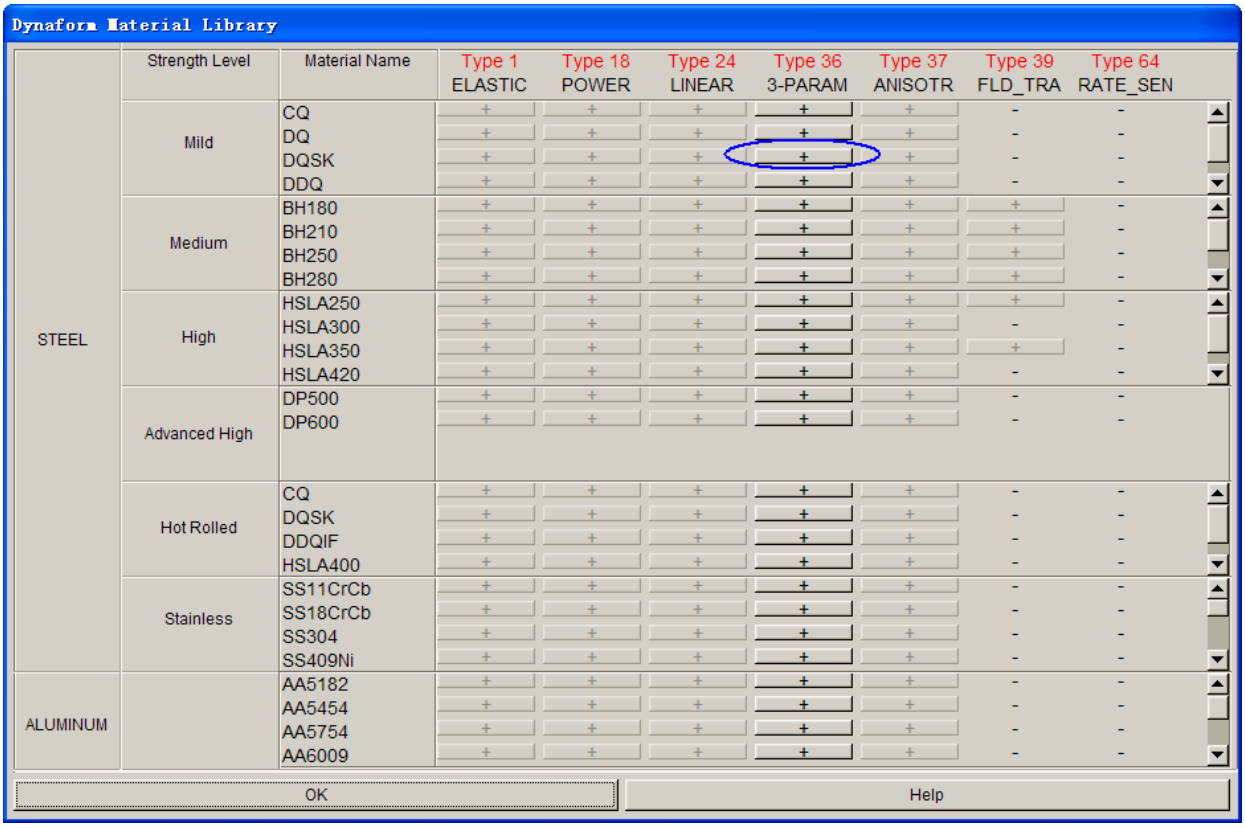


Figure 3.17: Material library window

6) Click “OK” to return to the Material dialogue window, then user may find the selected material type has already been added to Material List. See figure 3.18.

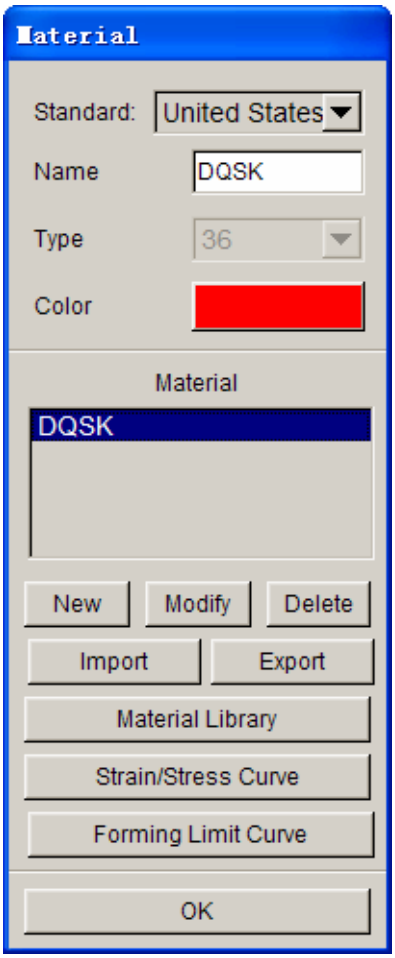


Figure 3.18: Define Material dialog box

7) Click “OK” to return to the Define Blank dialogue window，then user may find the None button behind Material has been changed into the selected material type name of DQSK, which indicates the part has been assigned a material type . See figure 3.19.

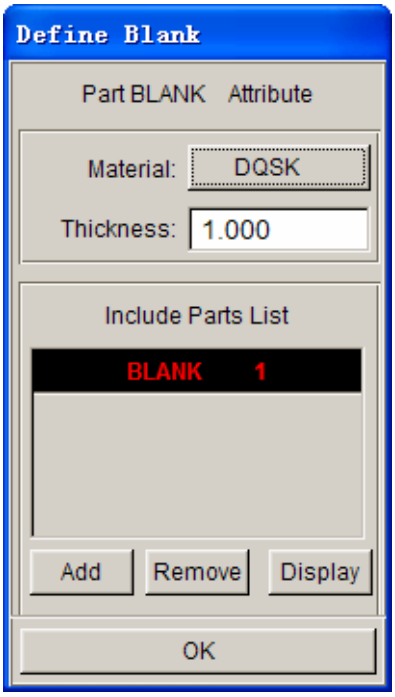


Figure 3.19: Define Blank dialog box

8) Then select the edit window of Thickness, define a sheet blank thickness is 0.9mm, as shown in the following figure 3.20.

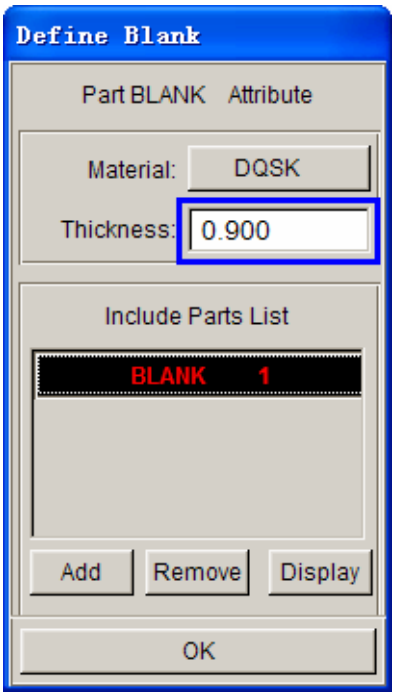


Figure 3.20: Define Blank dialog box

Note: *in MSTEP module, the simulated object is the finished components, but the defined material thickness is the initial sheet blank thickness.*

9) Click “OK’ in the Define Blank dialogue window to return to the MSTEP dialogue window, user can find the outline representing the sheet metal in the dialogue window has changed from red into green, which indicates the define of sheet-metal tool has been completed. See figure 3.21.

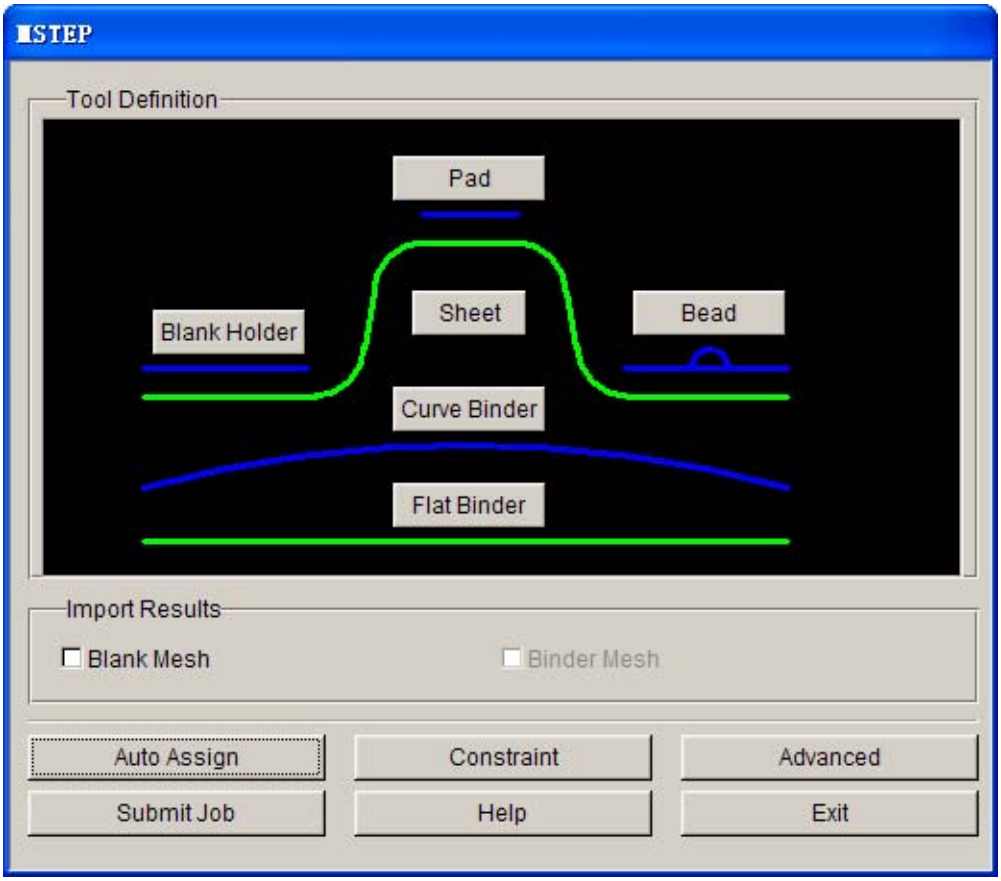


Figure 3.21: MSTEP GUI after sheet definition

10) Select the Blank Holder in the MSTEP dialogue window, then pop up a Define tools dialogue window, select “ADD” button, add the corresponding part to the blank holder tool.

11) Select “SELECT PART” in the pop-up DEFINE TOOL dialogue window as shown figure 3.32, then pop up the Define Ring dialogue window, click “Add”. See figure 3.33.

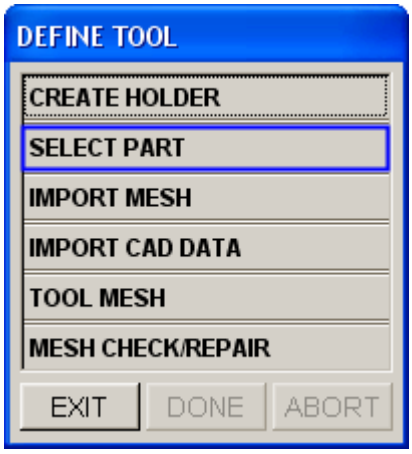


Figure 3.22: Define Tool dialog box

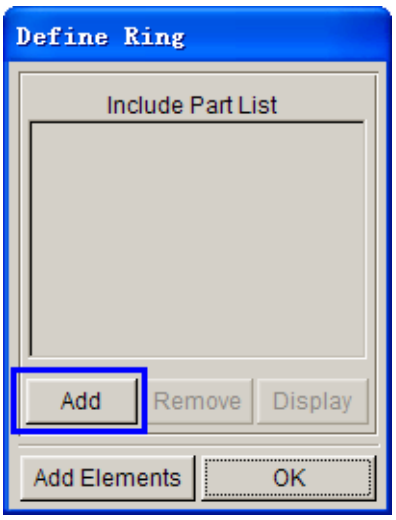


Figure 3.23: Define Ring dialog box

12) In the pop-up Select Part dialogue window, select the HOLDER part, then it is correspond to the blank holder tool, as shown in the following figure 3.24.

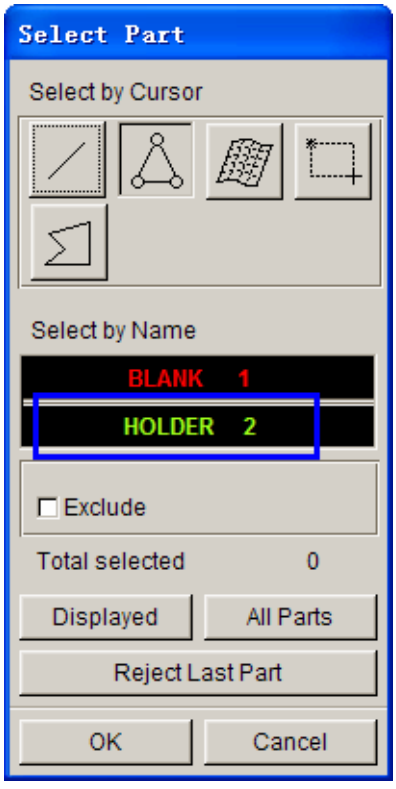


Figure 3.24: Select Part dialogue box

13) Click “OK” to return to the Define Ring dialogue window, user can find the selected target part has been added to the Include Part List. See figure 3.25.

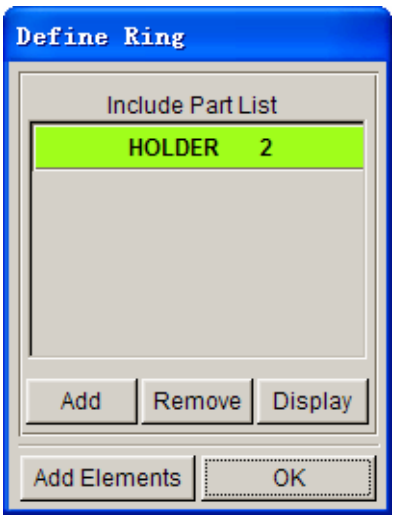


Figure 3.25: Define Ring dialog box

14) Click “OK” to return to the MSTEP dialogue window, user can find the outline representing the blank holder in the dialogue window has changed from red into green, which indicates the definition of blank holder tool has been completed. See figure 3.26.

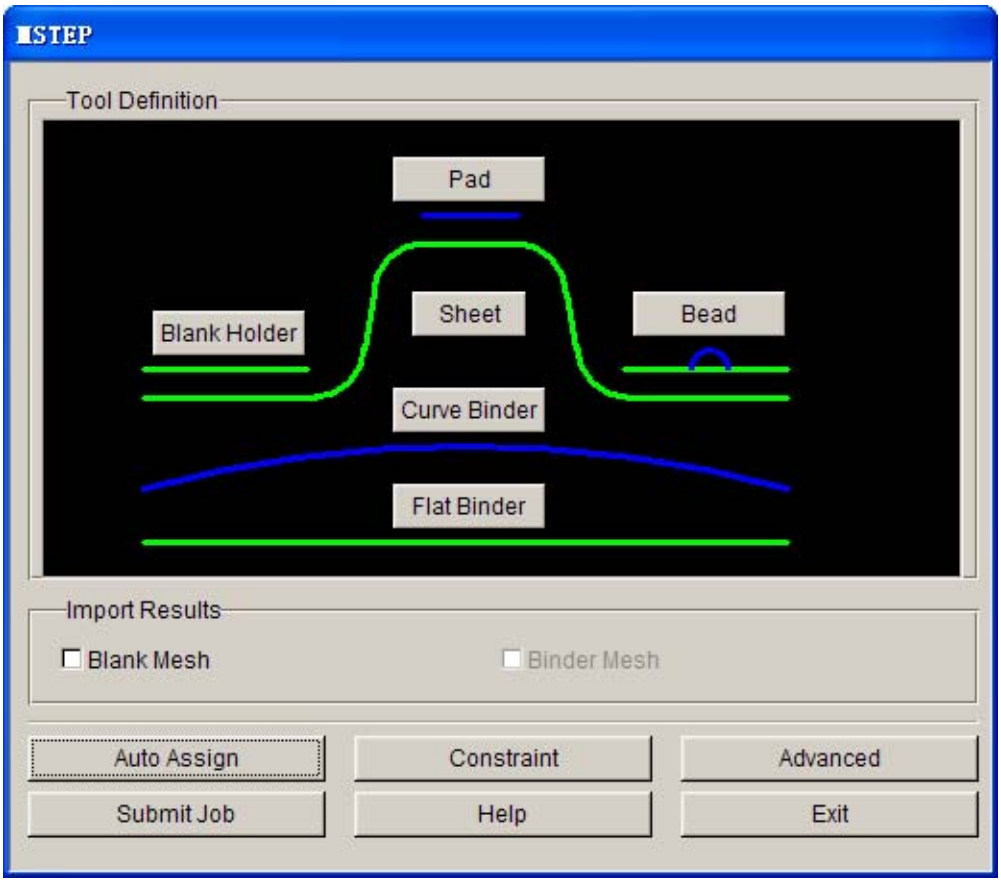


Figure 3.26: MSTEP GUI after Blank Holder definition

4. Define constraint

1) It's necessary to set symmetry constraint, since taking only 1/2 component to analysis in this example. Select the Constraint button in the STEP dialogue window to pop up the Constraint dialogue window, then define the symmetry constraint. See figure 3.27.

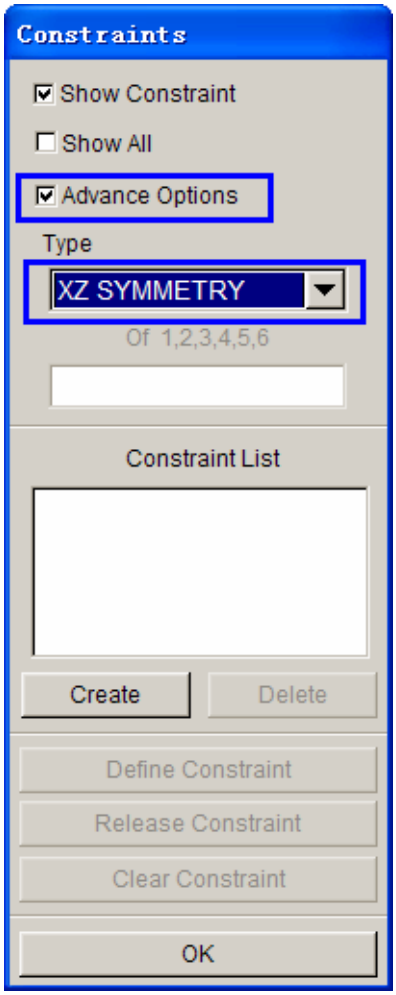


Figure 3.27: Define Constraints dialog box

2) Select the type of XZ SYMMETRY from the constraint type list "Type", which indicates the part is symmetry about the XZ plane. Click "Create', then pop up the dialogue window of SPC SET NUMBER, click "OK" to accept the default value. See figure 3.28.

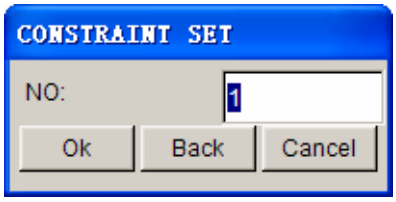


Figure 3.28: CONSTRAINT SET dialog box

3) Select the mode of Drag Window in the pop-up Select Node dialogue window, and pick the symmetry axis nodes by dropping mouse (only select the nodes belonging to the BLANK

part），as shown in the following figure 3.29.

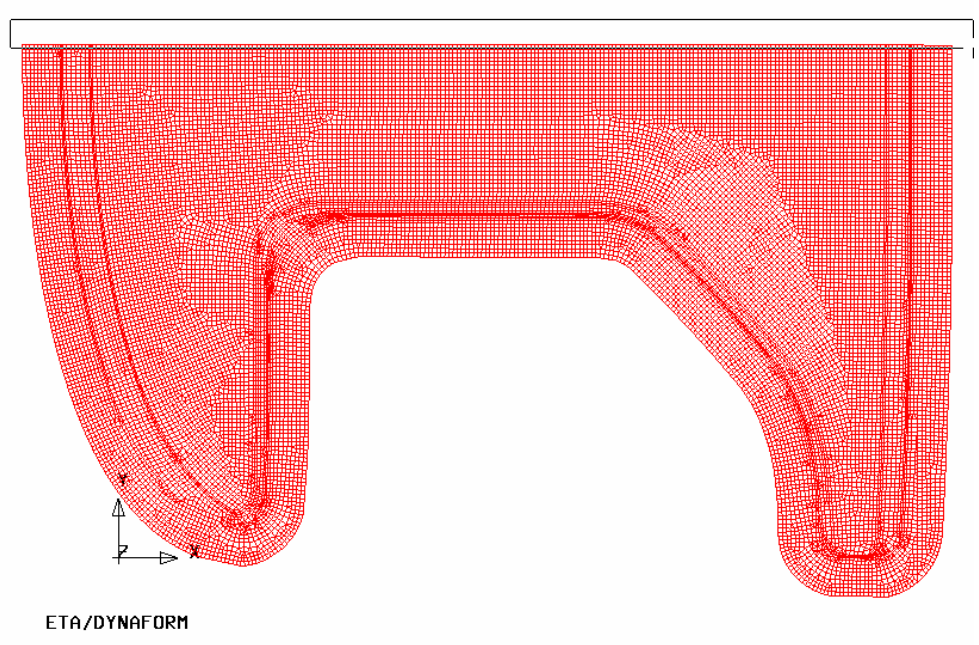


Figure 3.29: Illustration of the model by Drop Window

4) Return to the Constraint dialogue window, the defined constraint set number has added to Constraint List, and at the same time the defined constraint set member's symbol will be highlighted in the drawing display area, as shown in the following figure 3.30~3.31.

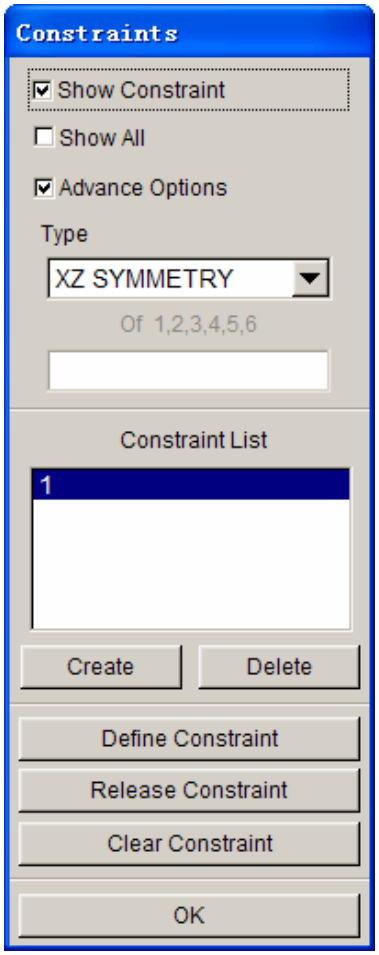


Figure 3.30: Define Constraints dialog box

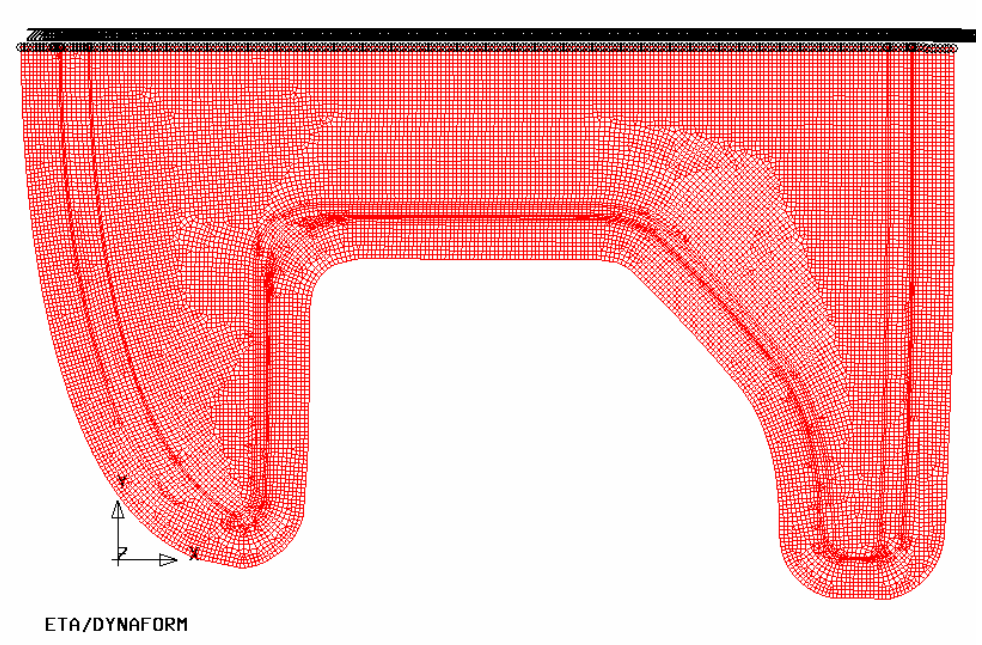


Figure 3.31: Illustration of constraints definition

5. define simulation parameter

1) Select “Advanced” in the MSTEP dialogue window to pop up the MSTEP SETTING dialogue window, then set the solver related parameters. See figure 3.32.

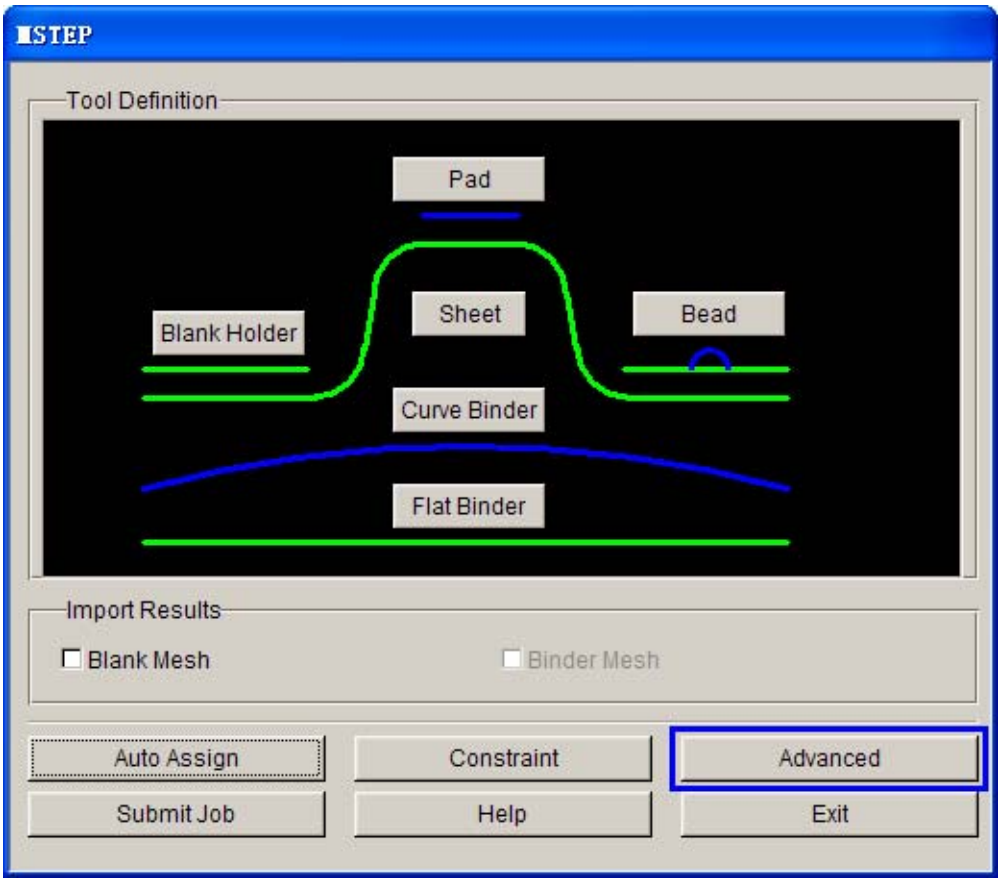


Figure 3.32: MSTEP GUI

2) The MSTEP module provides two solution modes: Accurate and Fast. Wherein the Accurate mode considers the impacts of process parameters in solution, such as the binder hold force, pad hold force and draw bead etc, and considers the impact of material parameters and plasticity deformation factors accurately, which is perfectly consist with the real formability process. The Fast mode ignores the impact of above parameters in solution, in order to speed up the computation, which correspond the elasticity unfold mode. In this example, the impact of blank holder tool should be considered, so select the Accurate mode, and set the binder hold force at 500,000 Newton and accept other default values, the result is shown in the following figure 3.33.

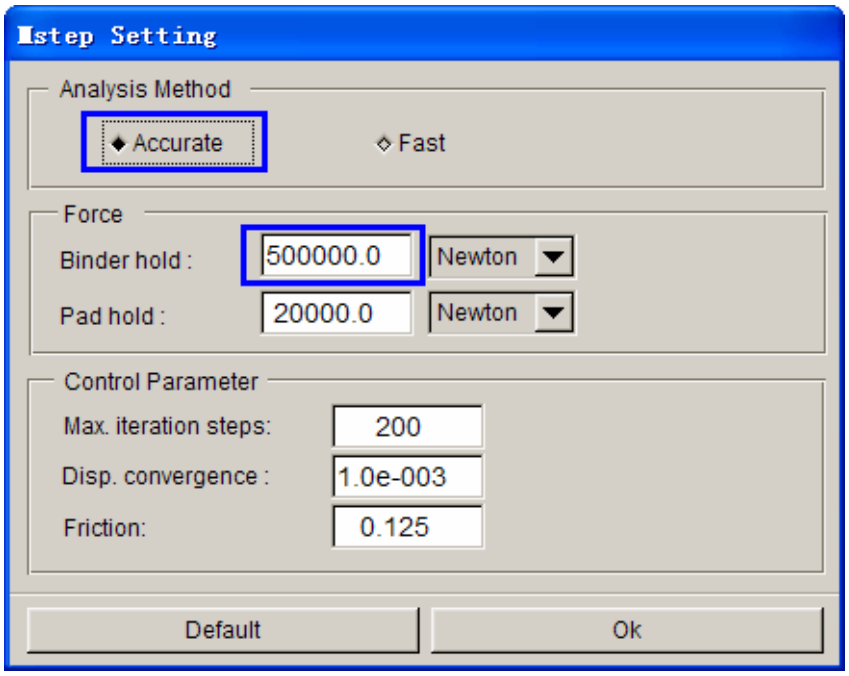


Figure 3.33: MSTEP Setting dialog box

3) Click “OK”, return to the MSTEP dialogue window.

6. start up the MSTEP solver

Now, in this example, all the solution related parameters have been defined, and then select the Submit Job button in the MSTEP dialogue window to start up the MSTEP solver. The solution operation is performed. See figure 3.34.

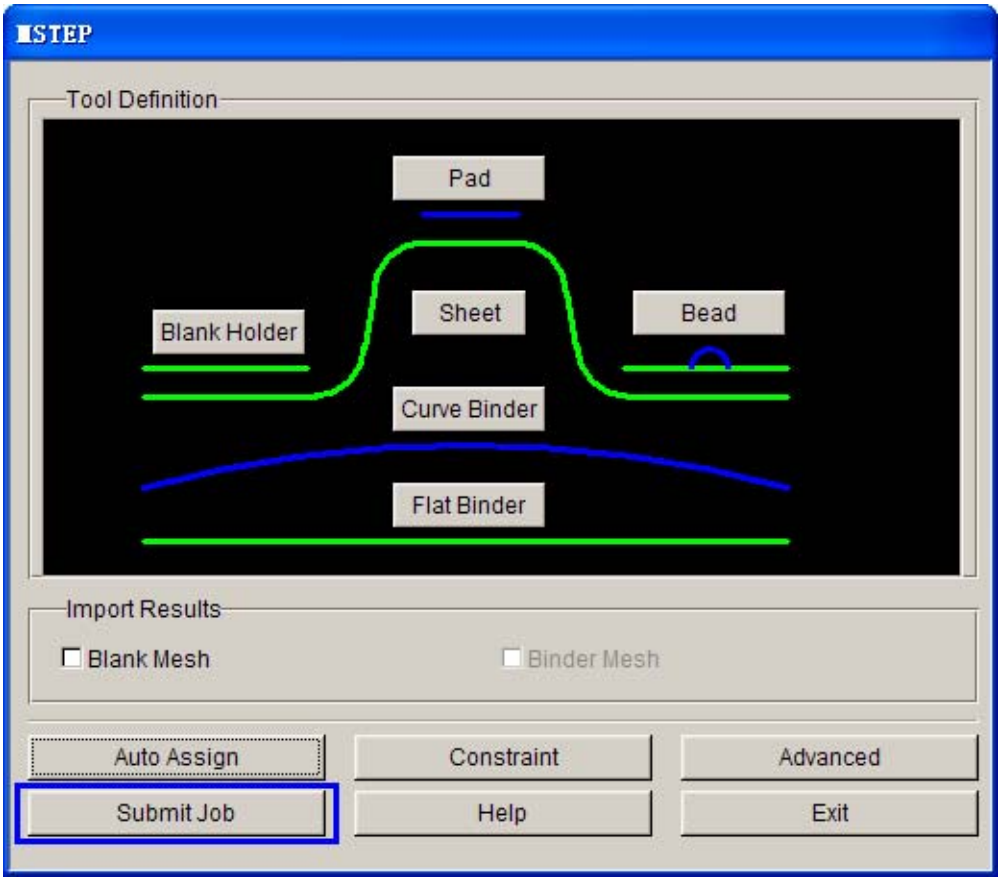


Figure 3.34: MSTEP GUI

V. Start up Post-Process and Analyze Simulation Result

After the MSTEP evaluation is finished, DYNAFORM will automatically read in the computed sheet blank unfold outline, and add it to a new part, the blue closed line shown in the following figure 3.35 is the part's sheet blank unfold outline.

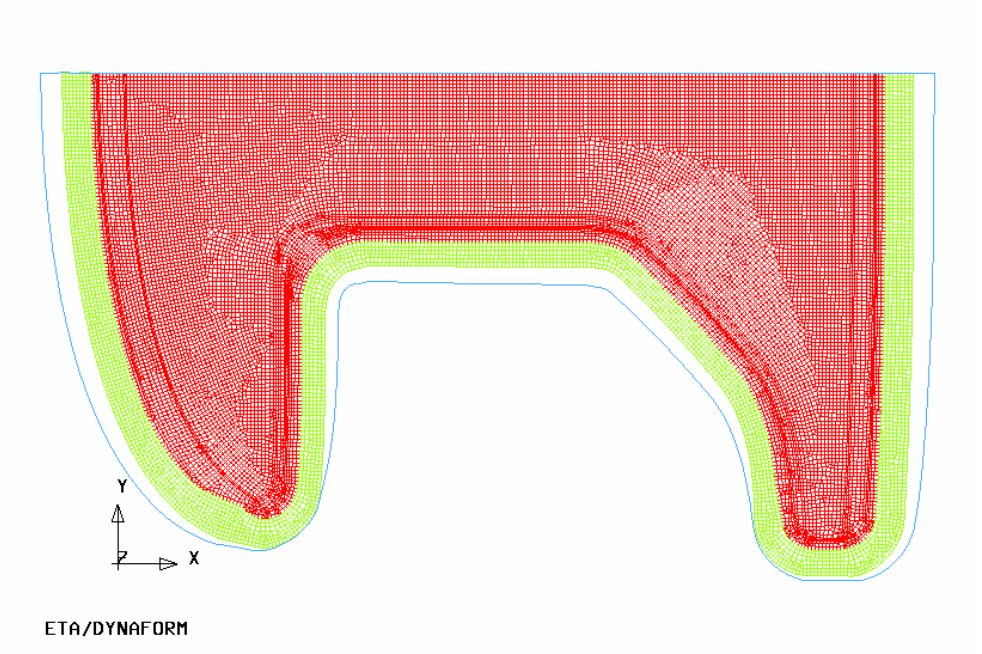


Figure 3.35: Illustration of sheet blank unfold outline

In order to observe the full information about the result, User may start up the Post-process program to analysis the result. In the result from MSTEP module, the emphasis should be laid on observing the forming limit diagram (FLD) and the thickness distribution diagram to check the flow and formability defects of sheet blank.

1. Select the PostProcess menu to start up the ETA/Post-Processor.

2. select menu **File→Open** as shown figure 3.36 or click the OPEN icon 

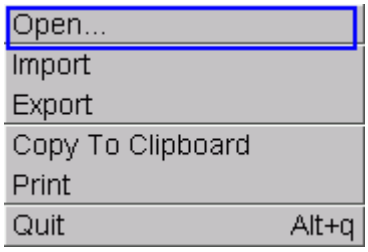


Figure 3.36: Open menu

3. Pop up the dialogue window to open file, select the file of dynain.mstep, and click “Open”, then the result file will be read in, which is shown in the following figure 3.37~3.38.

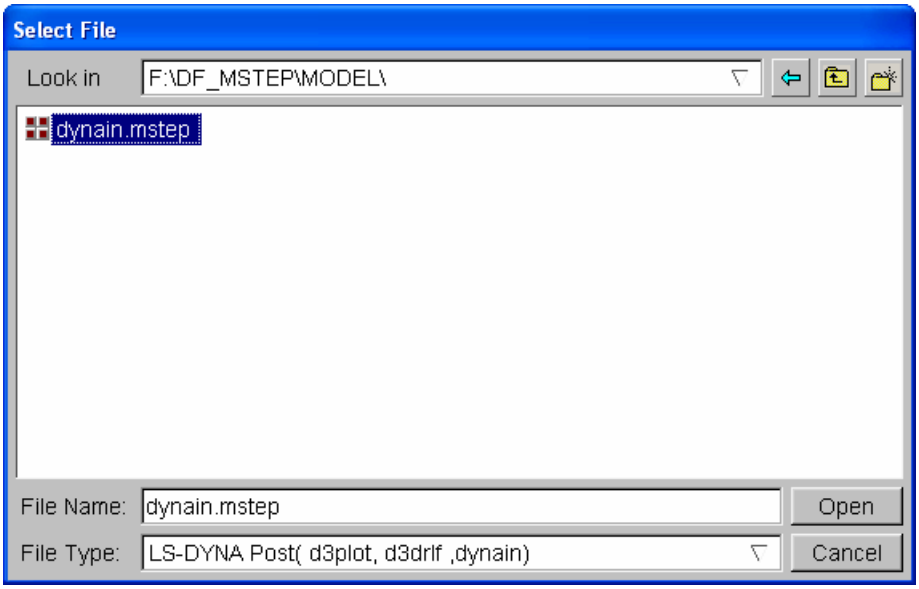


Figure 3.37: Open file dialog box

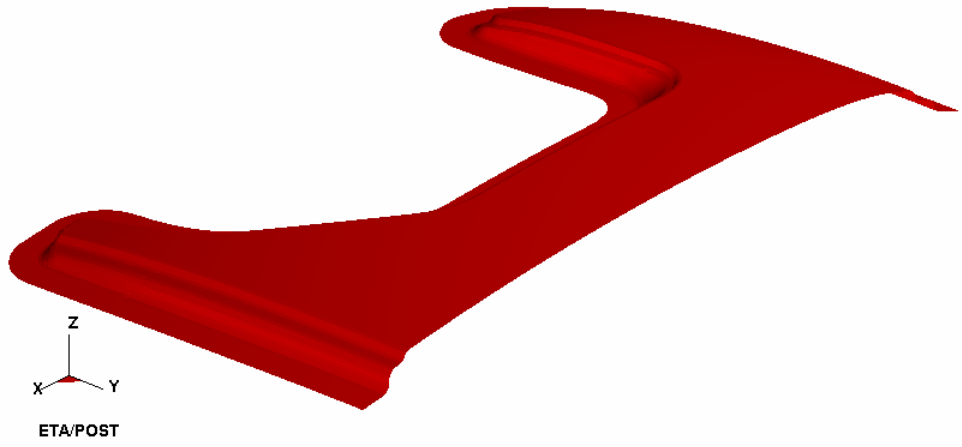


Figure 3.38: Illustration of dynain.mstep

4. forming limit diagram

   1) Select “FLD” in the result operation menu.

   2) Select “Middle” in the pull-down menu of **Current Component.** See figure 3.39.

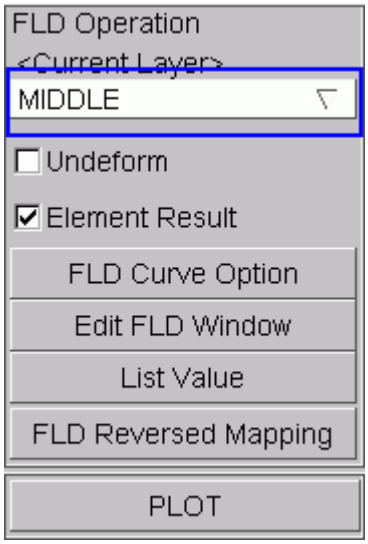


Figure 3.39: FLD dialog box

3) Click "**FLD Curve Option"** to set FLD parameters (n， t， r etc.).

4) Select "**Edit FLD Window"** to define the position of plotting FLD.

5) Click "PLOT" to plot the forming limit diagram of the formed component, as shown in the following figure 3.40.

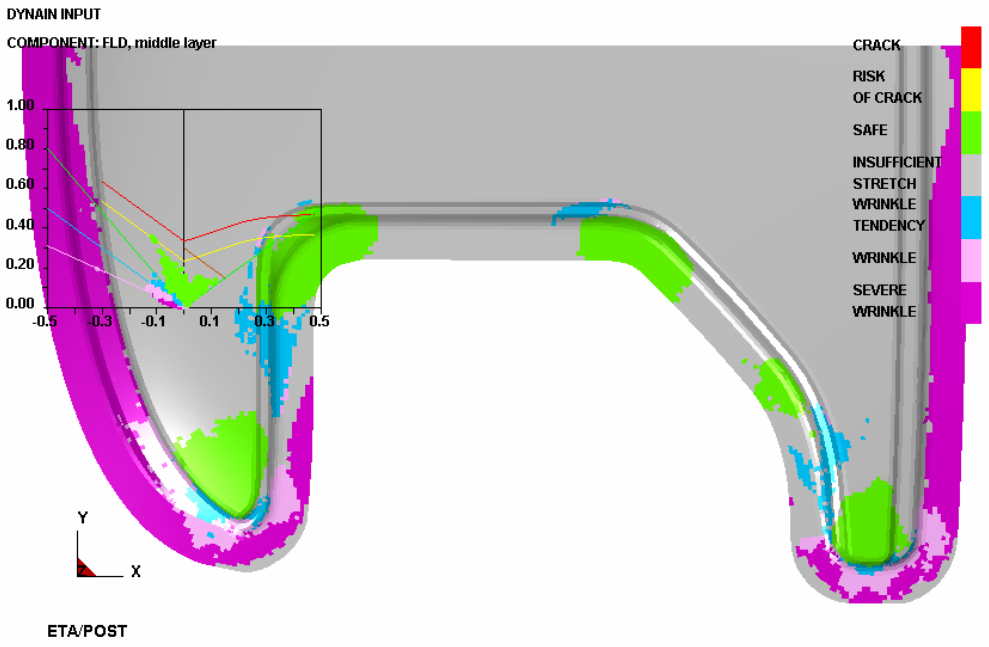


Figure 3.40: Illustration of FLD result

In this example, user may know from the FLD analysis that the binder hold force of 500,000 Newton can't fully satisfy the condition of component formability, so the process parameters of draw bead should be set to make the component shaped. The detail operation about adding the draw bead will be

introduced in the following chapters.

5. thickness animation / thinning animation as the following figure 3.41.

Figure 3.41: Thickness/thinning icon

1) Select "**Thickness"** from the result operation menu. See figure 3.42.

2) Select THICKNESS (absolute value) or THINNING (relative thinning rate) arbitrarily from the drop down menu of **Current Component**.

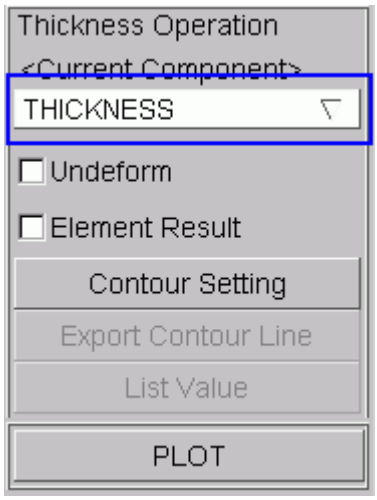


Figure 3.42: Thickness/thinning dialog box

3) Click "PLOT" to plot the thickness contour diagram, which is shown in the following figure 3.43.

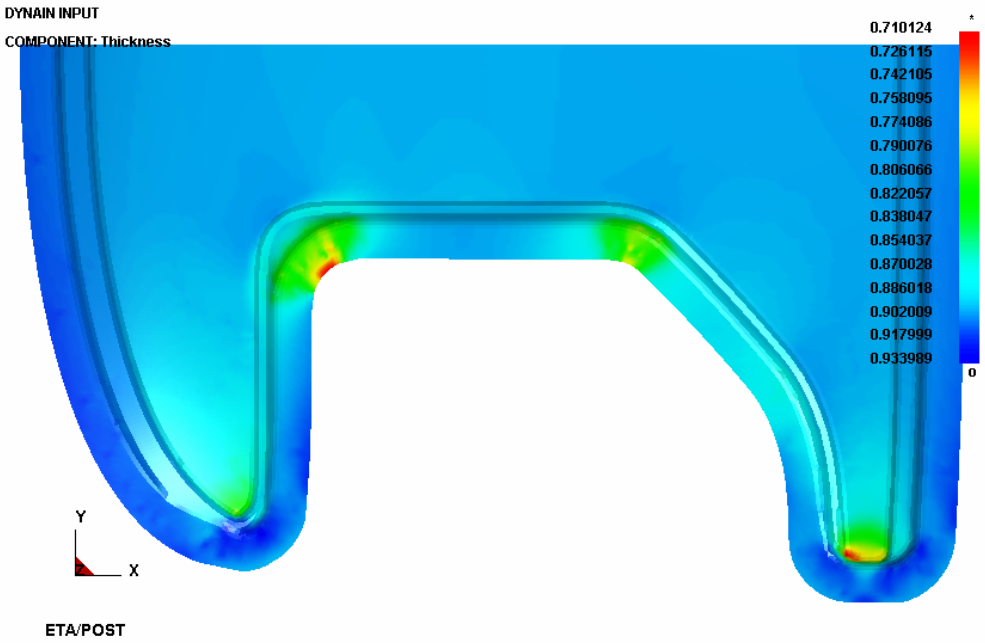


Figure 3.43: Illustration of thickness result

6. material flow path

1) Select menu **File→Import** to open the Import dialogue window. See figure 3.44.

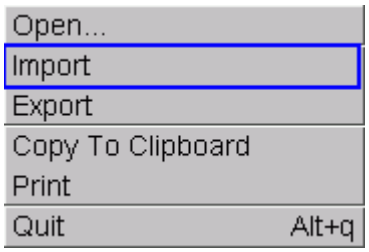


Figure 3.44: Import file menu

2) Select the file of MSTEP_model1_mstep.lin, click “Open”, and the sheet blank outline is read in, as shown in the following figures 3.45~3.46.

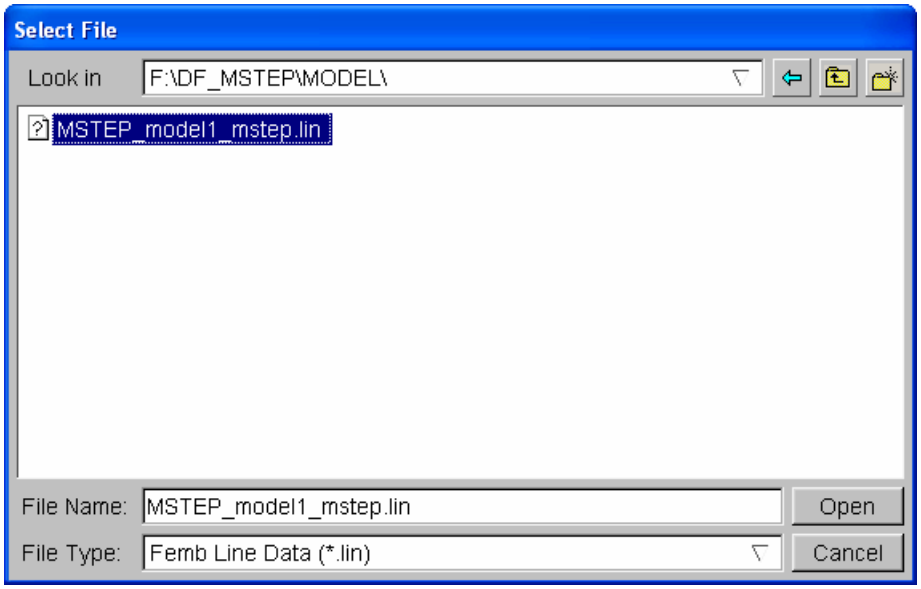


Figure 3.45: Open file dialog box

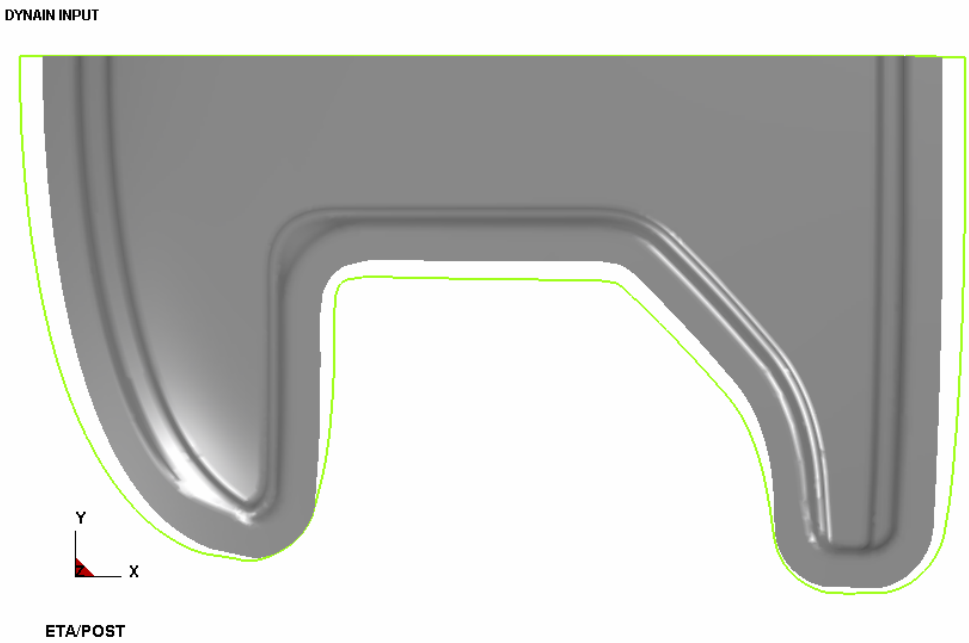


Figure 3.46: Illustration of MSTEP_model1_mstep.lin

VI. Process Parameter Optimizing

User may understand from the result analysis that the main problem in this example is the insufficiency of formability, so the process parameters should be further optimized to make the component shaped sufficiently. The binder hold force had been set at 500,000 Newton in above evaluation, so user may define draw bead force to make the component shaped fully.

1. Close the ETA/Post-Processor, and automatically return to the DYNAFORM pre-processor interface as shown in the following figure 3.47.

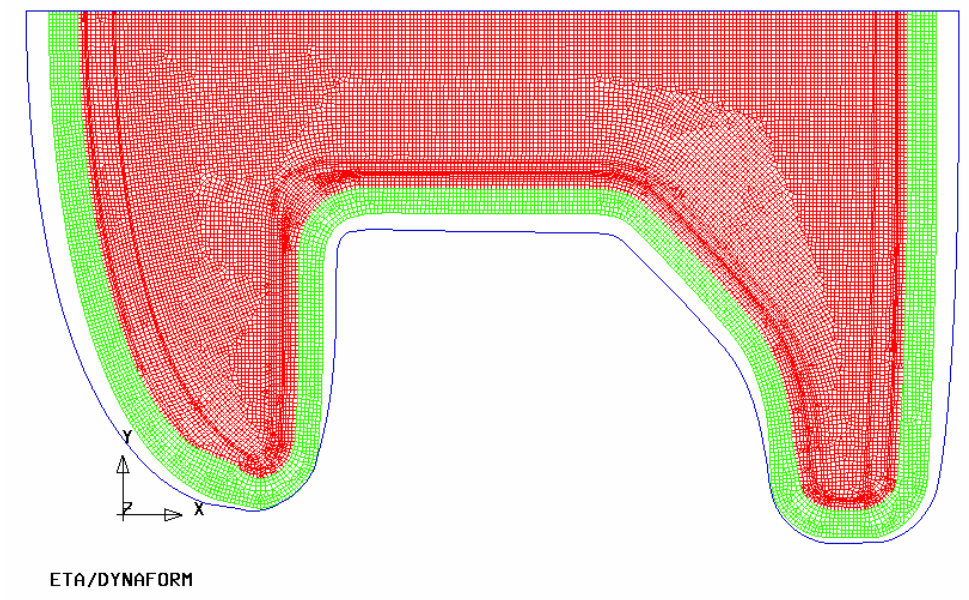


Figure 3.47: Illustration of the result

2. Select menu **File→Import** as the following figure 3.48or click the IMPORT icon 

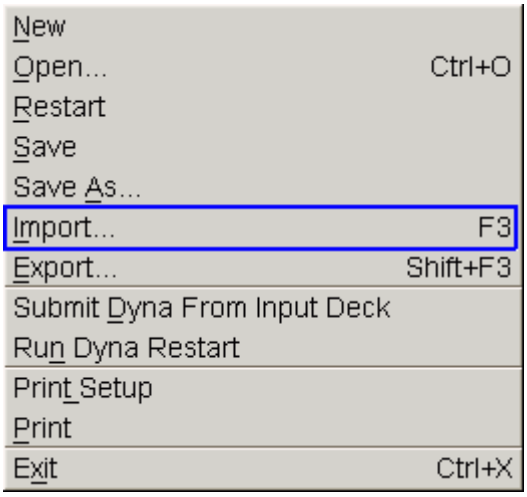


Figure 3.48: Import menu

Choose the directory of training file, select the file of **MSTEP**_model2_beadline.igs to import into the database. See figure 3.49.

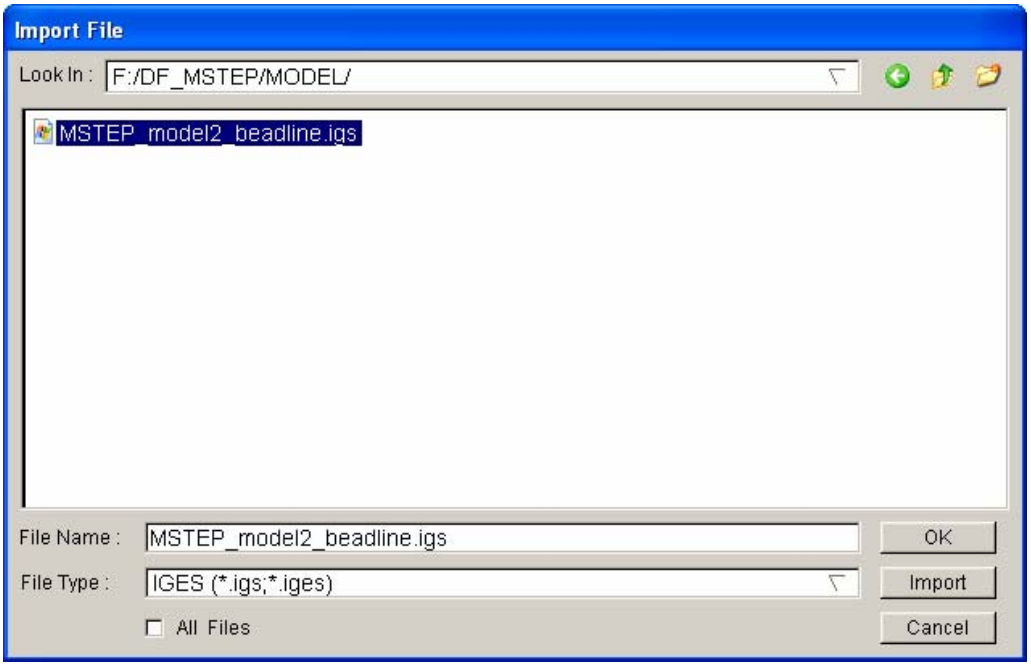


Figure 3.49: Import file dialog box

After importing the file, close the nodes and elements, and check out the content displayed in the screen to make sure it is the same as the curve model shown in the following figure 3.50, where in the purple curve is the base curve of the draw bead.

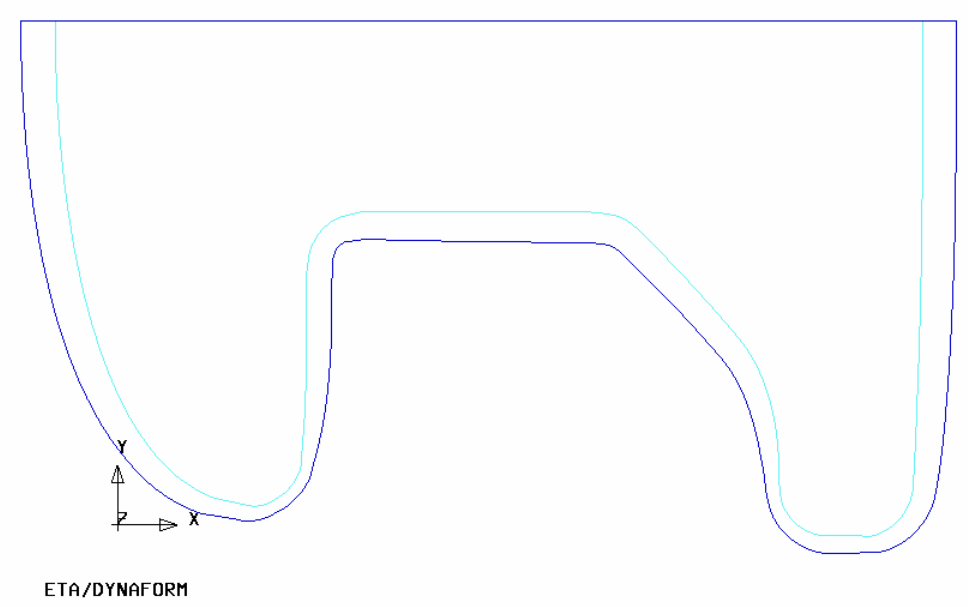


Figure 3.50: Illustration of the curve of draw bead

3. Select menu **BSE→MSTEP** to open MSTEP solution module. See figure 3.51.

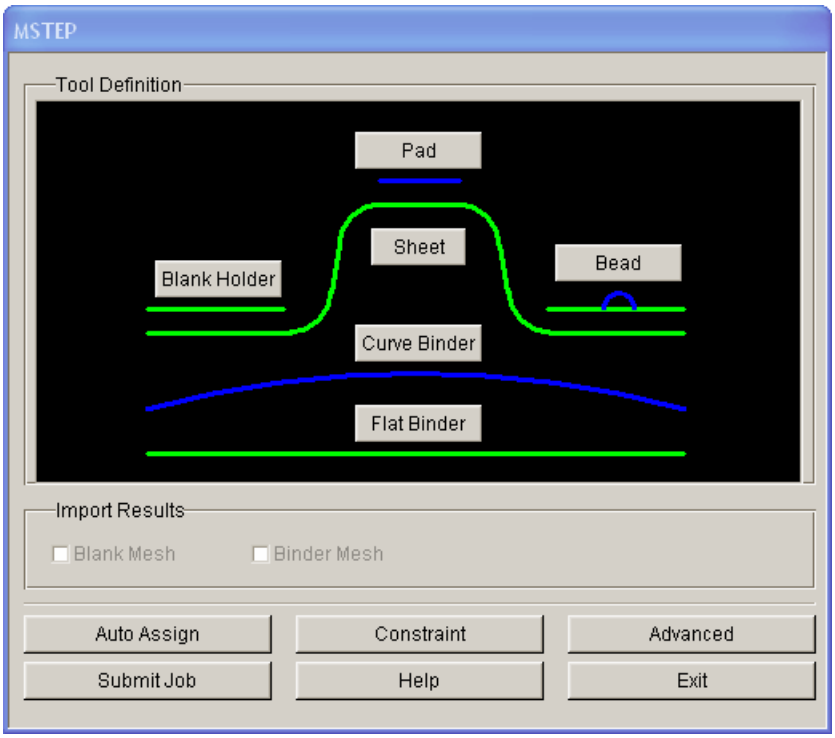


Figure 3.51: MSTEP GUI

4. define draw bead

   1) Select “Bead” from the MSTEP dialogue window, then pop up the Drawbead dialogue window, and select “New” to pop up DRAW BEAD PROPERTIES dialogue box. User can accept the default setting as the following figure 3.53. Click OK button, then a new drawbead property is

added to the list of Drawbead Property.

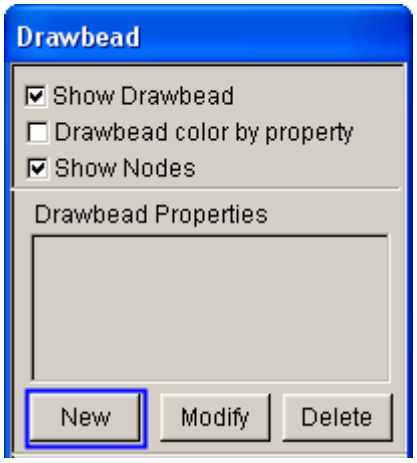


Figure 3.52: Define Drawbead property dialog box

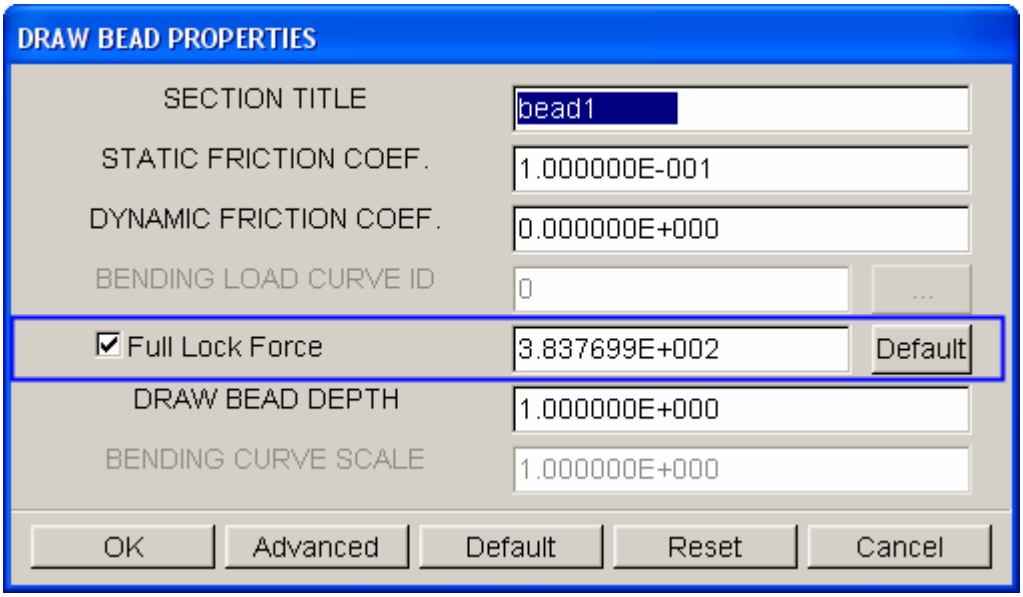


Figure 3.53: DRAW BEAD PROPERTIES dialog window

2) Then press New button as shown figure 3.54 to create new draw beads.

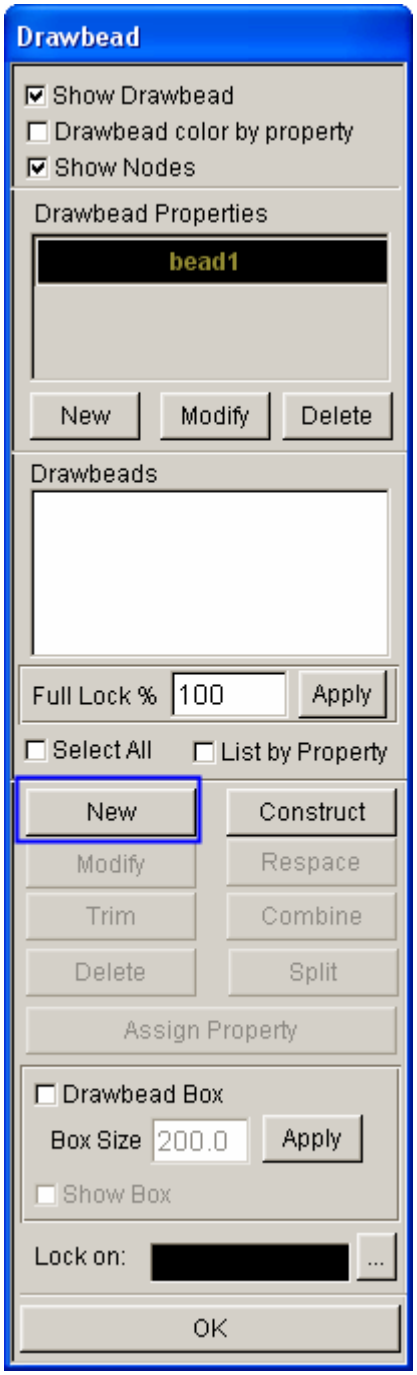


Figure 3.54: Define Drawbead dialog box

3) Pop up the Select Line dialogue window, then pick the draw bead's base curve imported above, the system will show the selected curve, as shown in the following figure 3.55.

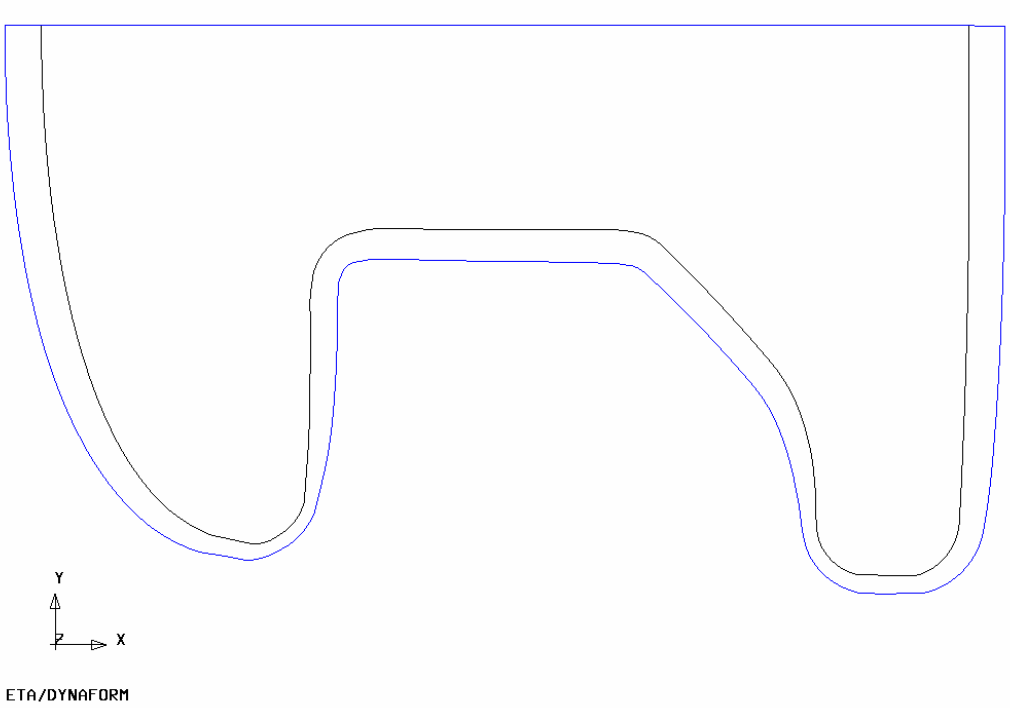


Figure 3.55: Illustration of selected curve

4) Click “OK” in the Select Line dialogue window to pup up DrawBead Input dialogue window, user can select Chordal Deviation option and acceptt the default setting in the dialogue window as shown in the following figure 3.56.

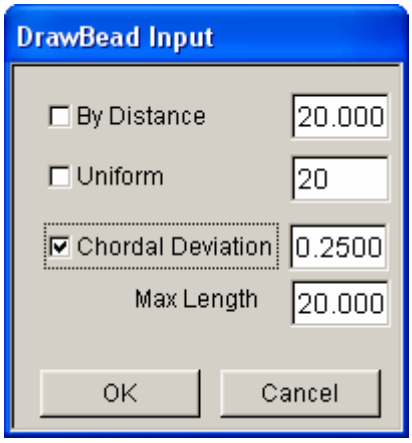


Figure 3.56: DrawBead Input dialog box

5) New drawbeads named DrawBead1 and DrawBead2 are added to the list. Then use mouse cursor to pick DrawBead1 from the drawbead list. Type in 50 (%) in the input data field of Full Lock% and press Apply button. DrawBead2 is the same to DrawBead1. See figure 3.57.

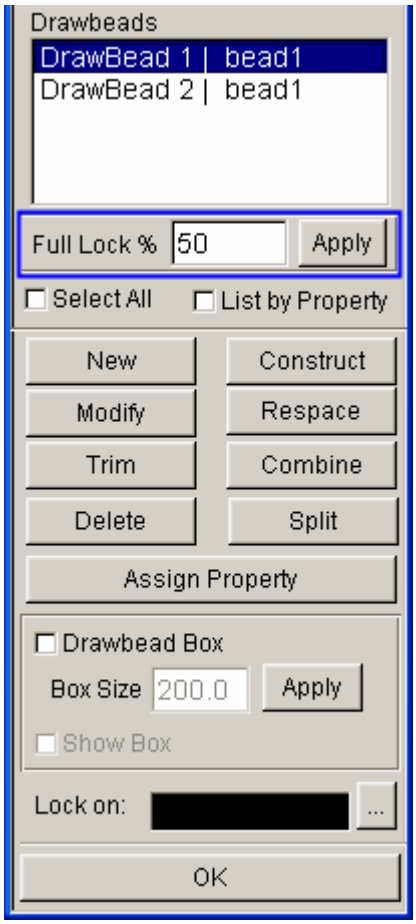


Figure 3.57: DrawBead dialog box

6) Click “OK” in the Define Drawbead dialogue window to return to the MSTEP dialogue window, user may find the outlines representing the draw beads change from red into green, which indicates the define of draw beads has been completed. See figure 3.58.

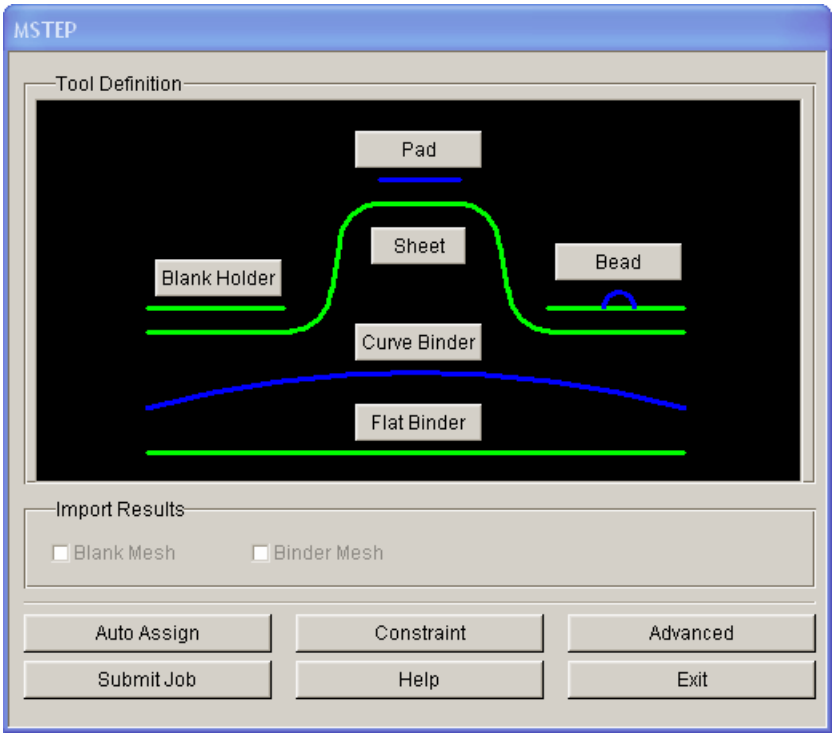


Figure 3.58: MSTEP of Bead definition

7) Repeat the “define simulation operation” above, select the solution mode of Accurate, set the binder hold force at 500,000 Newton, and accept the other default values.

5. Start up the MSTEP solver to perform the solution.

6. Once the MSTEP evaluation ends, DYNAFORM automatically read in the resulting blank unfold outline and add it to a new part. As shown in the following figure 3.59, the closed yellow curve is the blank unfold outline, to which the draw bead is added.

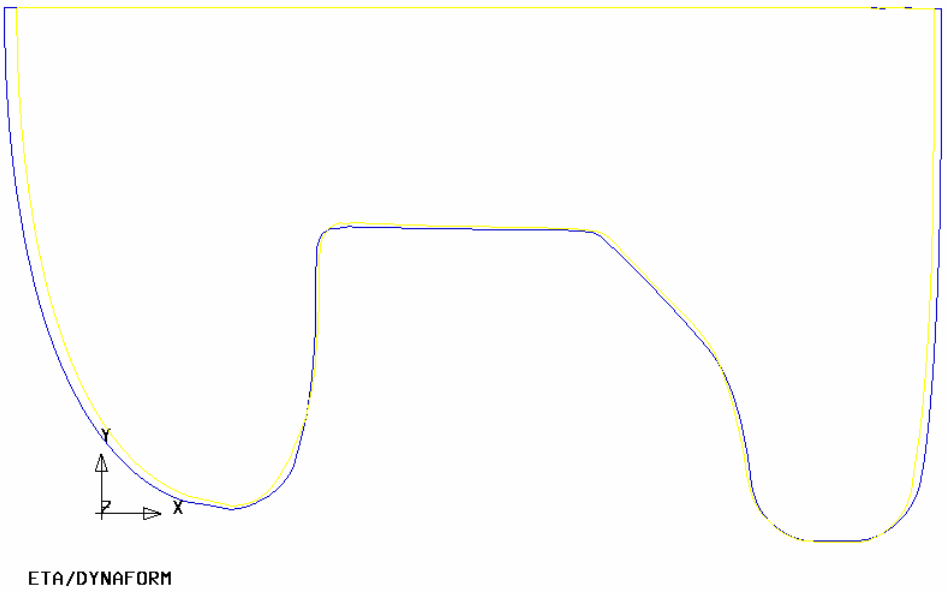


Figure 3.59: Illustration of blank unfold outline

7. Start up ETA/Post-Processor, open the dynain.mstep to observe the detail information about the result. See figure 3.60.

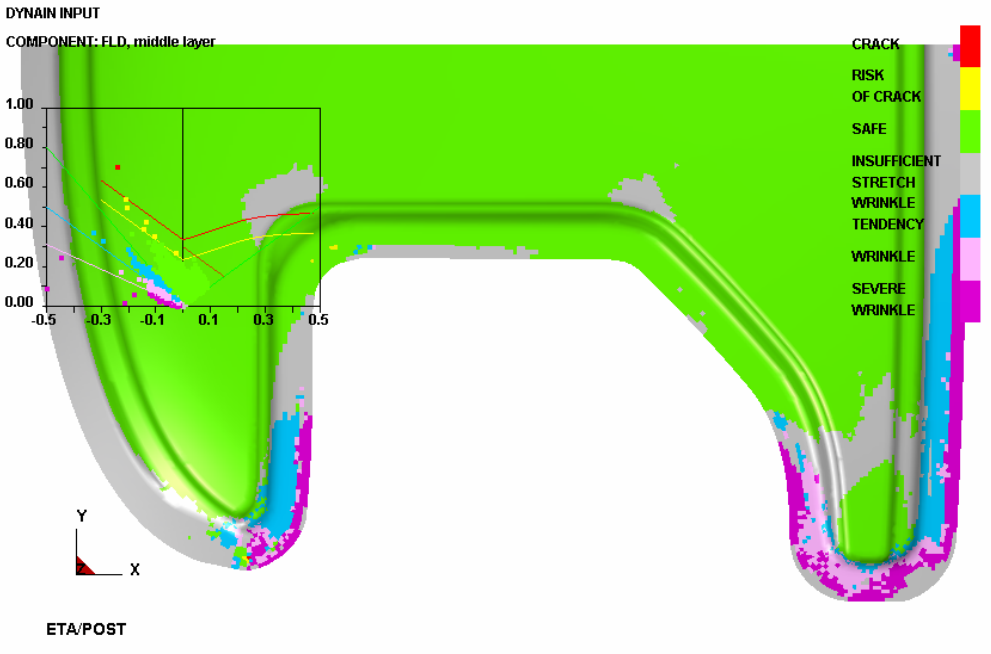


Figure 3.60: Illustration of FLD result

8. Observe the resulting FLD, user may find that the component shapes sufficiently after being added the draw bead parameters, which indicates all the process parameters meet the formability condition.

Example 4. Tailor Welded Blank Forming

I. Create Database and Read in Model File

Start up eta/DYNAFORM 5.5.

Workstation and Linux user may enter the command of "df55"（default）in the command line to start up DYNAFORM5.5. PC user may double click DYNAFORM 5.5（DF55）icon or select DYNAFORM from the program menu to start up it.

Once starting up eta/DYNAFORM, the program automatically creates the empty database file of Untitled.df in default. It's necessary for user to import CAD or CAE model to start working.

1. Select menu **File→Import** as shown figure 4.1or click the IMPORT icon 

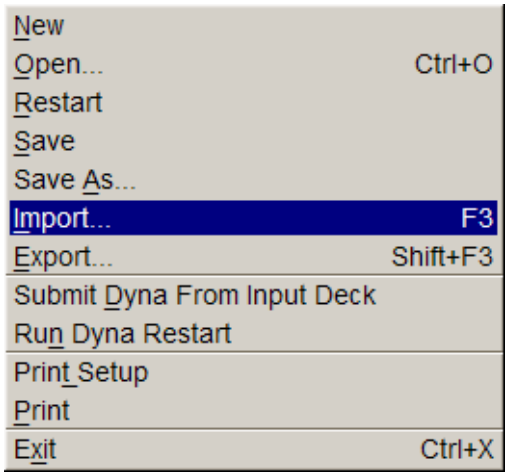


Figure 4.1: Import menu

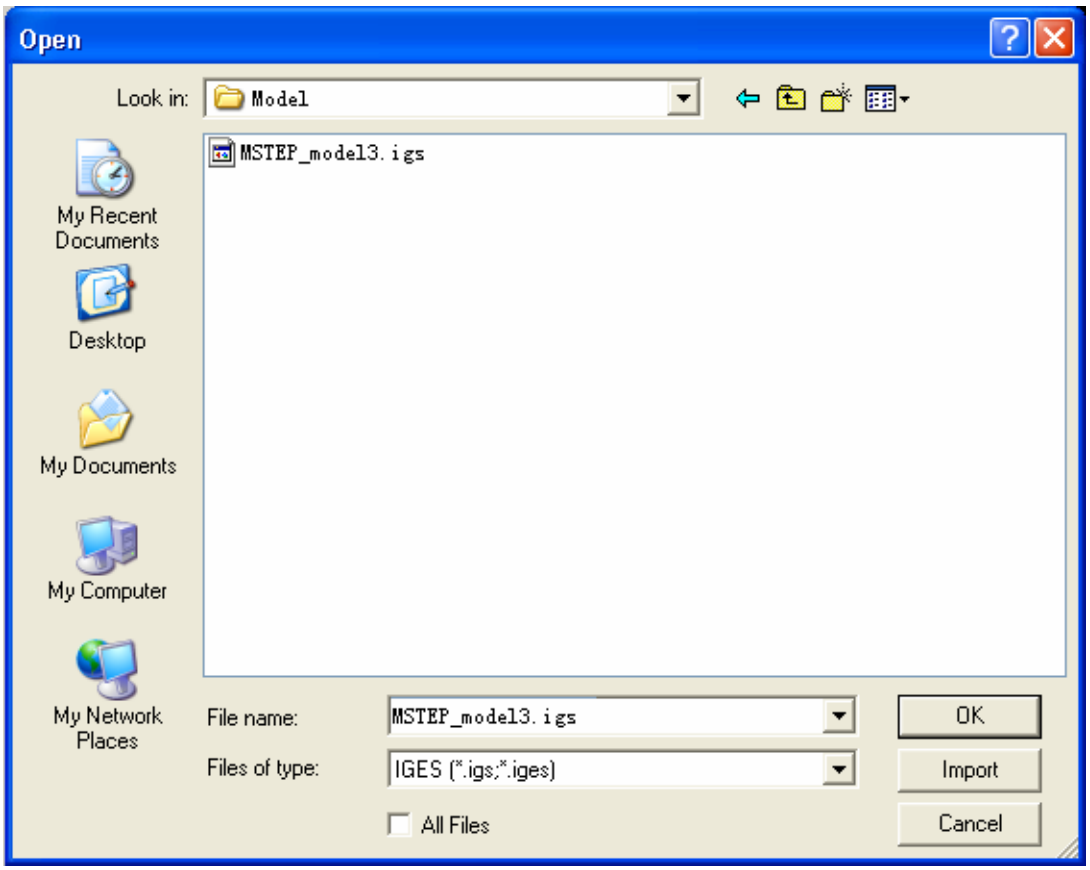


Figure 4.2: Open file dialog box

Find the directory of training file, and import the file of MSTEP_model3.igs into the database. See figure 4.2

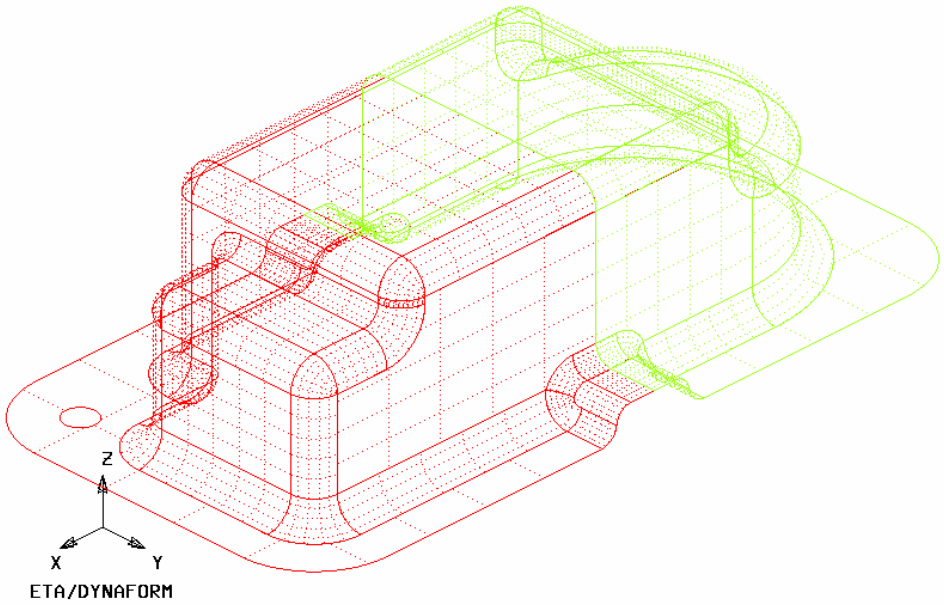


Figure 4.3: Illustration of MSTEP_model3.igs

After importing the file, check the displayed picture to make sure it consist with the model shown as follow figure 4.3. In default, the model is shown as an isometric view.

Note: *the icon is different in the different system platform; the other icons in the toolbar will be discussed in the following. User may refer to the eta/DYNAFORM Manual to understand all the functions of the toolbar.*

2. Save the database file in a specified working directory. Select menu **File→Save as** or click the SAVE icon  , input ”MSTEP_model3.df”, then select “save” to save it and exit the dialogue window. See figure 4.4.

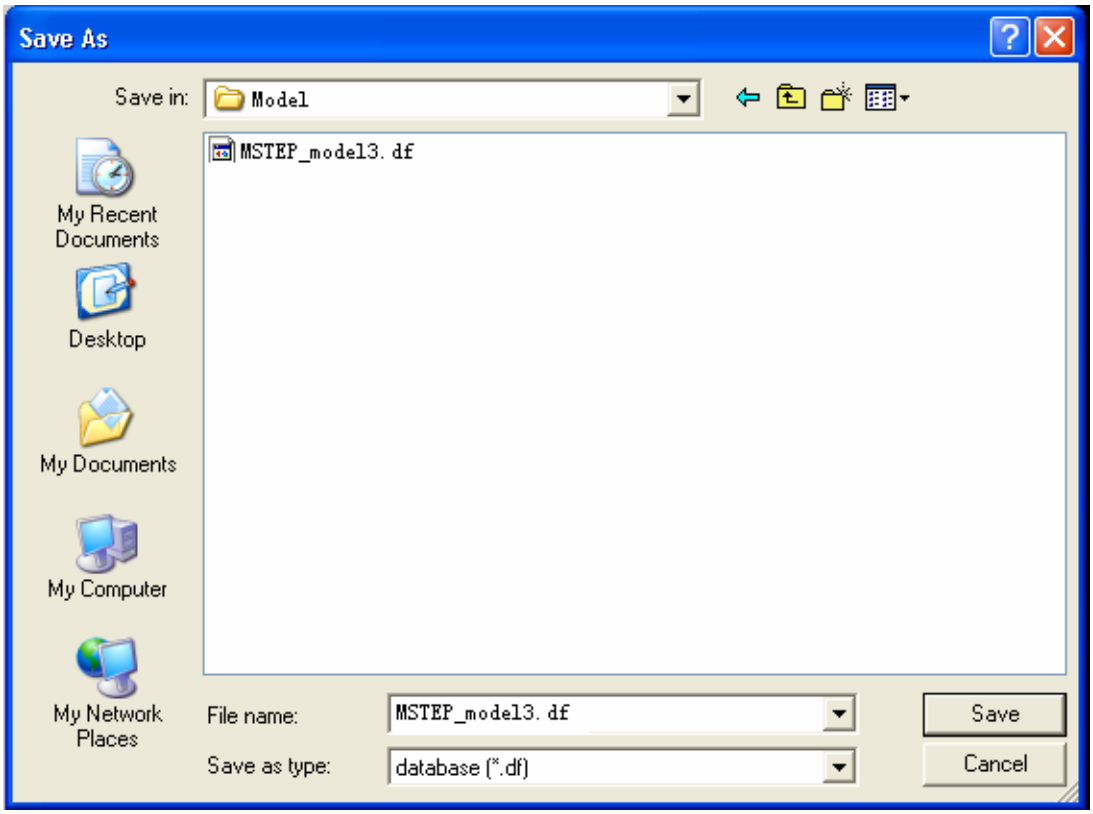


Figure 4.4: Save as file dialog box

User may refer to the eta/DYNAFORM Manual to obtain the detail information about the units and file type of DYNAFORM database.

II. Edit Parts in Database

In eta/DYNAFORM, all the models are managed based on the part. All the entities will be created or read in the part in default. User may refer to the Eta/DYNAFORM Manual to obtain the detail information about the operation of the part.

The functions of Edit Part are to edit the part property or delete the part. See figure 4.5.

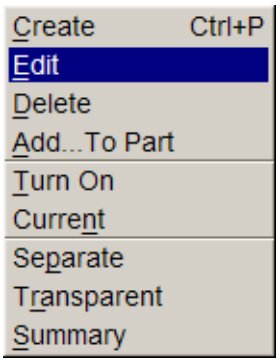
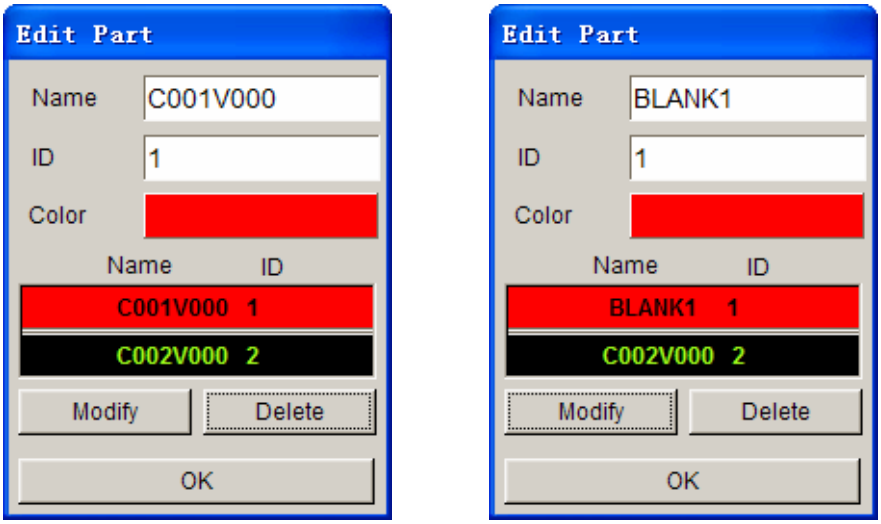


Figure 4.5: Edit menu

1. Select menu **Parts→Edit**, then pop up the Edit Part dialogue window. All the defined parts are displayed in the list and indicated by their name and ID. User may change the name and ID, and delete some parts form the database.

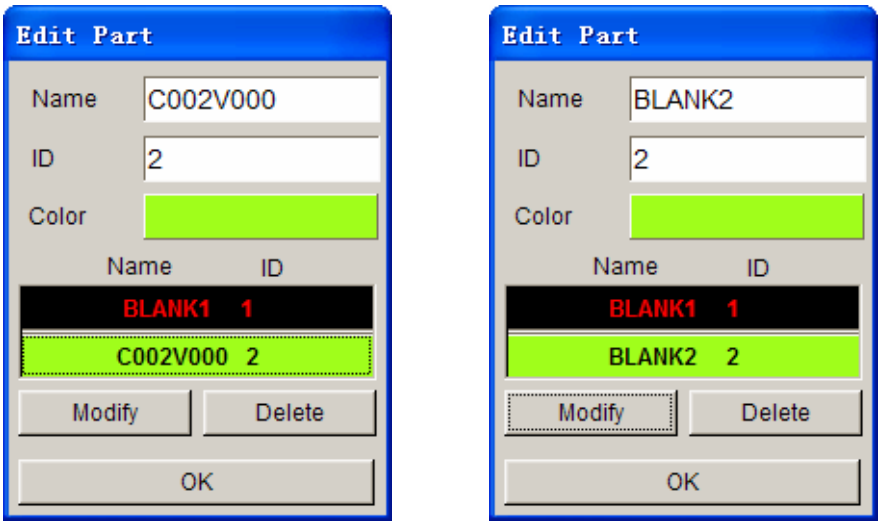
2. Select C001V000 from the part list. As shown in the following figures, user can input “BLANK1” in the input box behind Name and not change the part color, then click the Modify button in the down-left corner to affirm these edit. See figure 4.6.



a) b)

Figure 4.6: Edit Part dialog box

3. Using the same operation mode, change the part name of P2 into HOLDER, the result is as follow figure 4.7.



a) b)

Figure 4.7: Edit Part dialog box

4. Click “Close” to exit the edit operation.

III. Meshing

In the software of **eta/DYNAFORM**, most meshes use the term of Surface Mesh to be meshed. This function auto meshes the provided surface data, which is a fast and simply meshing tool.

- Create mesh

1. Select menu **Preprocess→Elements** as following figure 4.8 or use the shortcut key **Ctrl + E** to open the meshing menu.

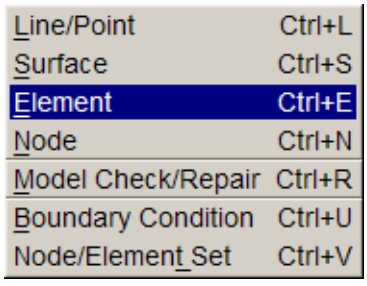


Figure 4.8: Element menu

2. Select the **Surface Mesh** from the Element menu as shown in the following figure 4.9.

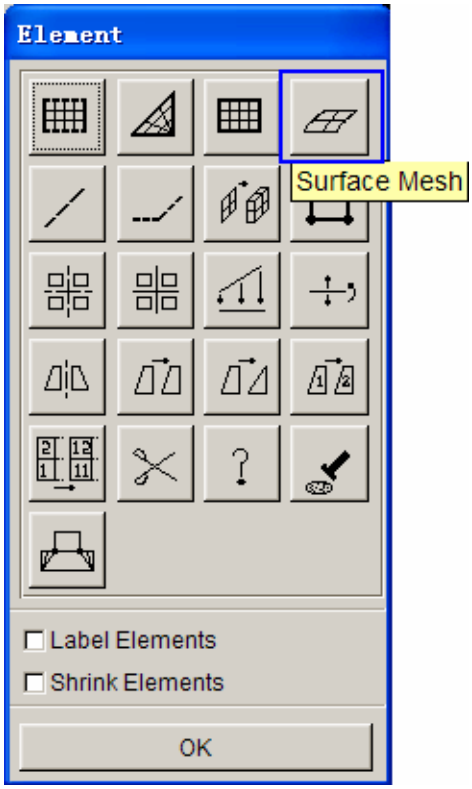
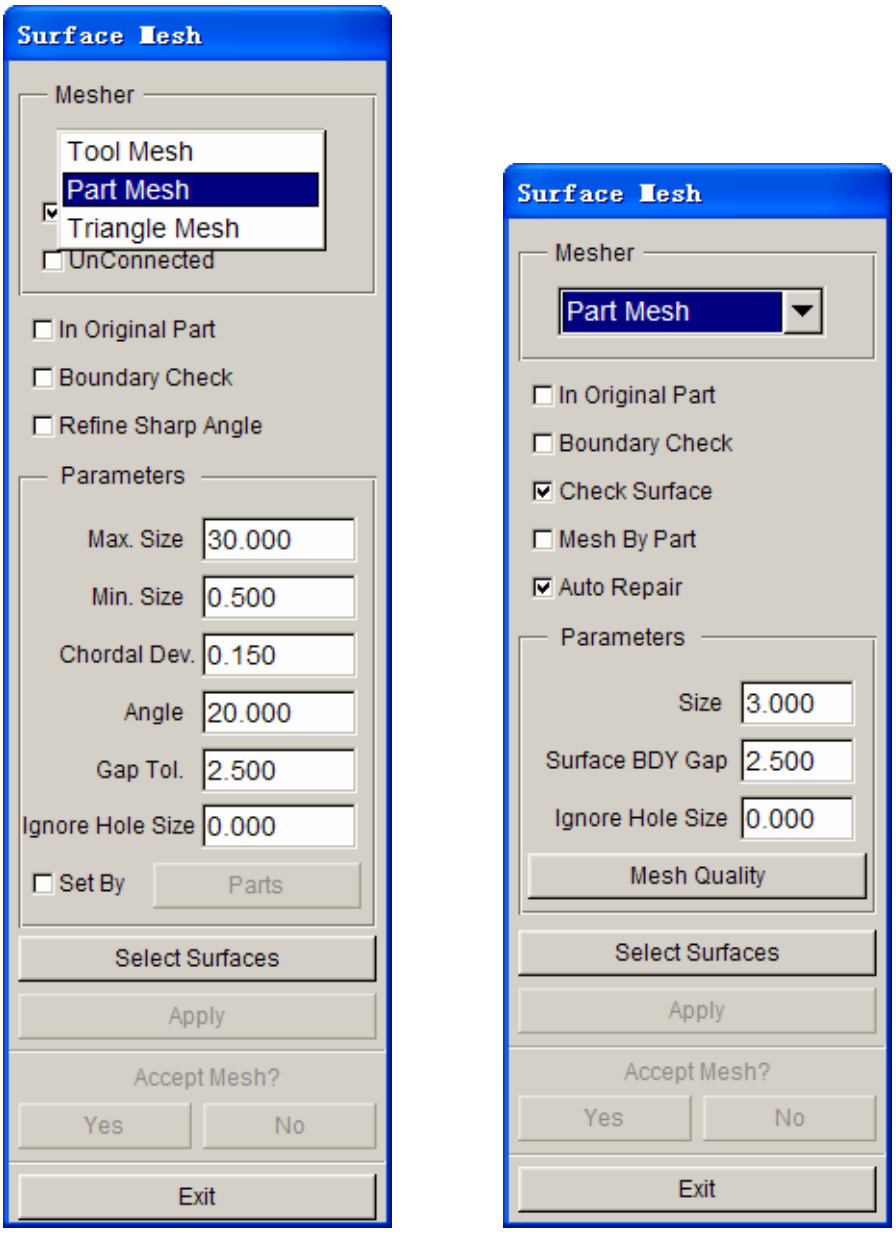


Figure4.9: Element dialog box

3. In the **Surface Mesh** dialogue window, select Part Mesh from the pull-down menu of Mesher, then a new Part Mesh dialogue window is popped up. See figure 4.10.



a) b)

Figure 4.10: Surface Mesh dialog box

Note: *In the BSE/Preparation/Part Mesh, the called surface mesh is the same as above, but the default set is Part Mesh.So it's unnecessary to change.*

4. Select "In Original Part", change the mesh size from 10.000 into 2.000, and keep the other terms in default, as shown in the following figure 4.11.

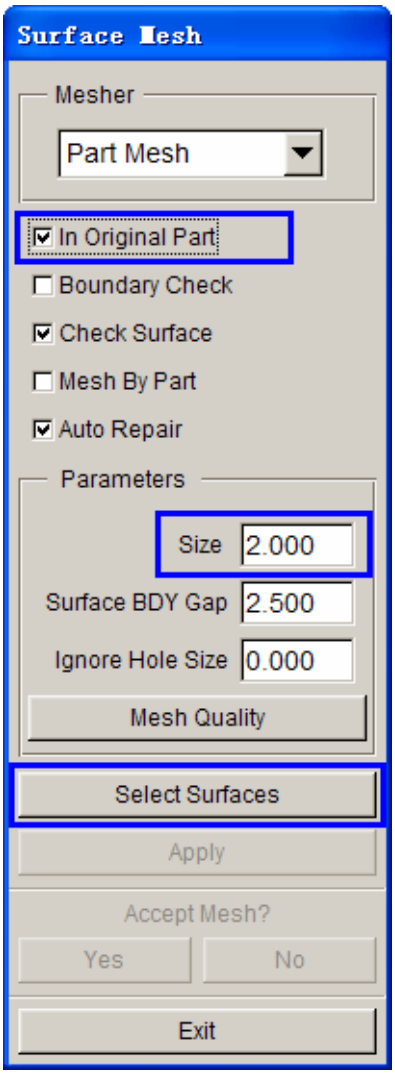


Figure 4.11: Surface Mesh dialog box

5. Select "**Select Surfaces"** from the **Surface Mesh** dialogue window. See figure 4.12.

6. Select "**Displayed Surf**" from the **Select Surface** dialogue window.

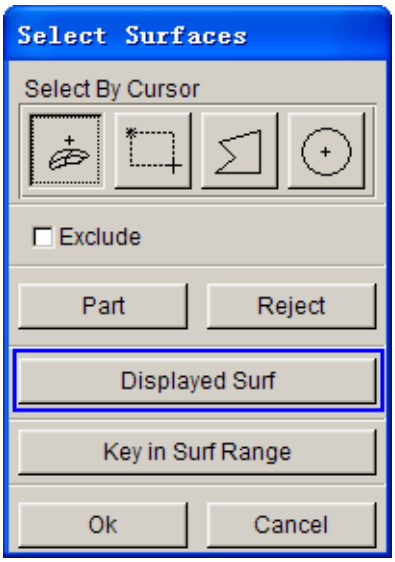


Figure 4.12: Select Surfaces dialog box

7. Pay attention to that the surfaces on display have turned white, which indicates all the surfaces are choose. There are some different methods to select the surfaces in the dialogue window; user may see every icon's function by putting the mouse on them.

8. Select "**Apply**" from the **Surface Mesh** dialogue window.

9. The generated meshes are displayed in white. When the system prompts "**Accept Mesh?**", select **"YES"**. User may compare the generated mesh to the following figure 4.13.

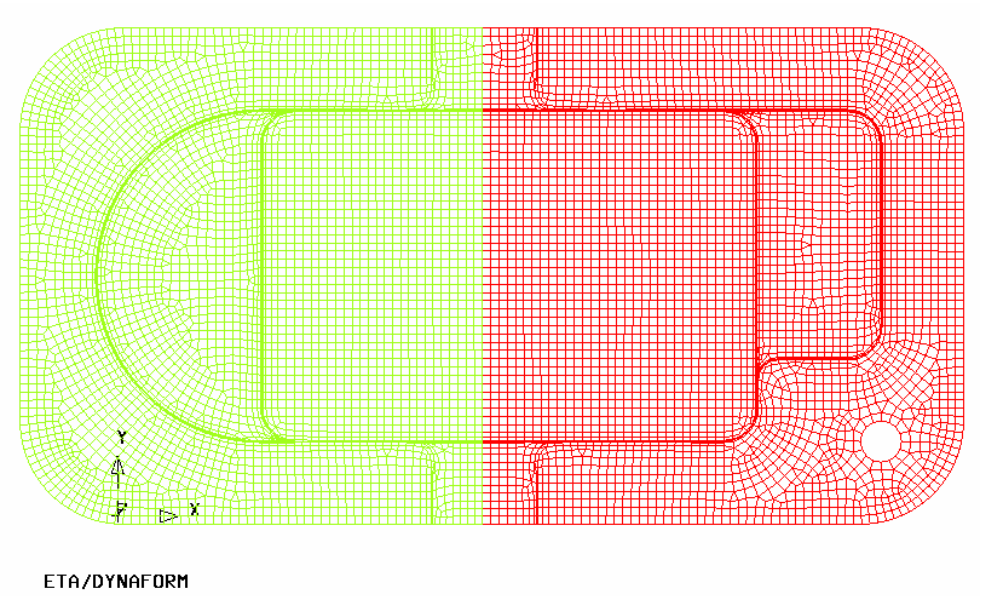


Figure 4.13: Illustration of mesh result

10. Select "**Exit**" from the **Surface Mesh** dialogue window to exit.

11. Save the database.

12. Now, all the meshing operations are completed. User may close the lines and surfaces by selecting “surface” and “Lines” from the down-right corner of the screen. This help to observe the resulting meshes. See figure 4.14.

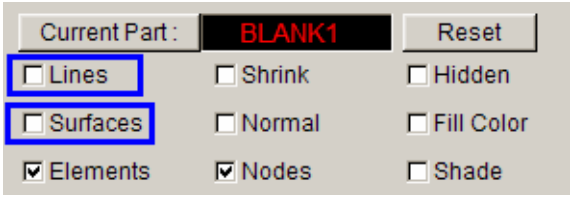


Figure 4.14: Display options

IV. Mesh Check

The meshes have been created. But it’s necessary to check the mesh quality, in order to prevent that the meshes have some potential defects impacting the simulation. All the tools used to check the mesh quality lie in the menu of **Preprocess→Model Check/Repair.** User may check the meshes by selecting menu **Preprocess→Model Check/Repair** or using the shortcut key **Ctrl + R**. See figure 4.15.

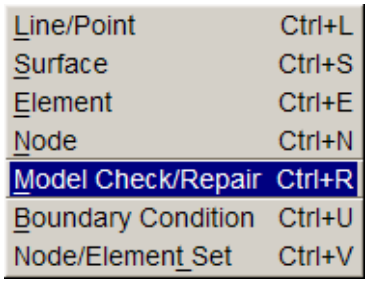


Figure 4.15: Model Check/Repair menu

Open the Model Check dialogue window, as shown in the following figure 4.16.

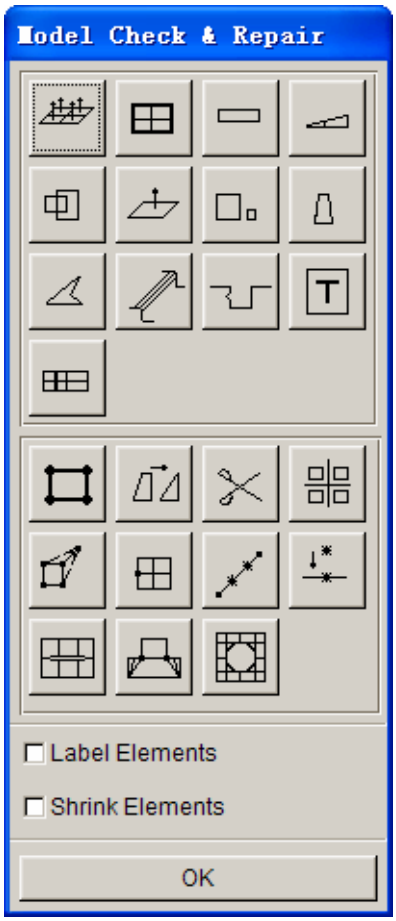


Figure 4.16: Model Check/Repair dialog box

As shown above, the Model Check dialogue window comprises some functions helping user to check the mesh quality. In this example, the used functions are mainly element check warpage, model boundary display, and auto plate normal. Other functions can be learned from the eta/DNAFORM manual.

- CHECK WARPAGE 

1. Select "**Warpage**", a new dialogue window is shown.
2. In the WARPAGE CHECK dialogue window, change the check standard from 5.0 degree (default) into 3.0 degree. See figure 4.17.

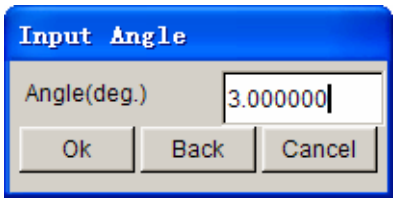


Figure 4.17: Input Angle dialog box

3. Select “OK” to check, all the failed elements will be highlighted, and a new dialogue window is popped up to ask if replace the failed quadrilateral elements with the triangular elements. See figure 4.18.

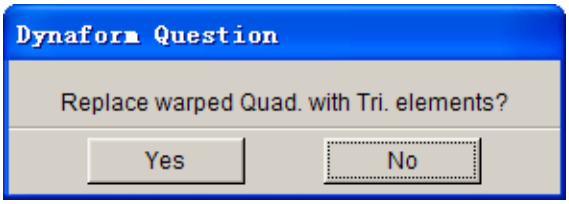


Figure 4.18: Dynaform Question dialog box

4. Select “YES”, then each of the failed elements is cut into two triangular elements. Select “NO”, then a new dialogue window is popped up to ask if include the failed elements to a new part. See figure 4.19.

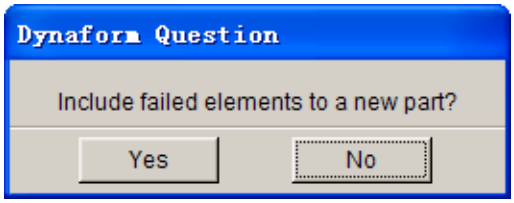


Figure 4.19: Dynaform Question dialog box

5. In this example, select the button of “YES” to cut all the failed elements into triangular elements.
6. Click “EXIT” to exit the CHECK WARPAGE.

- BOUNDARY DISPLAY 

This function allows the user to check the gaps and holes between the elements in the model, as well as the degradation elements. The eta/DYNAFORM highlights the boundaries of these defects, so that user may modify these defects manually.

1. Select **Model Check/Repair→ Display Model Boundary,** See figure 4.20.

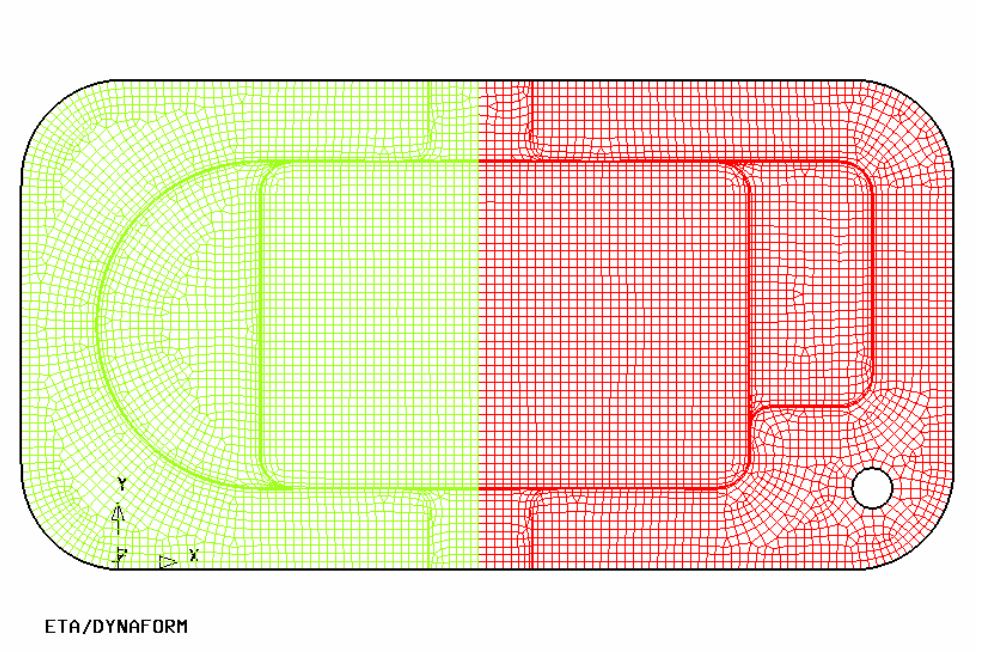


Figure 4.20: Illustration of model boundary

2. Close all the nodes and elements via the **Display Options** in the down-right corner of the screen, and only the highlighted boundaries are displayed. This make user check the little gap that can't be found easily when the meshes are displayed. The result from this example is as the following figure, only the edge lines and the inner hole boundary is highlighted, so the mesh model passes the check and needn't be modified. See figure 4.21.

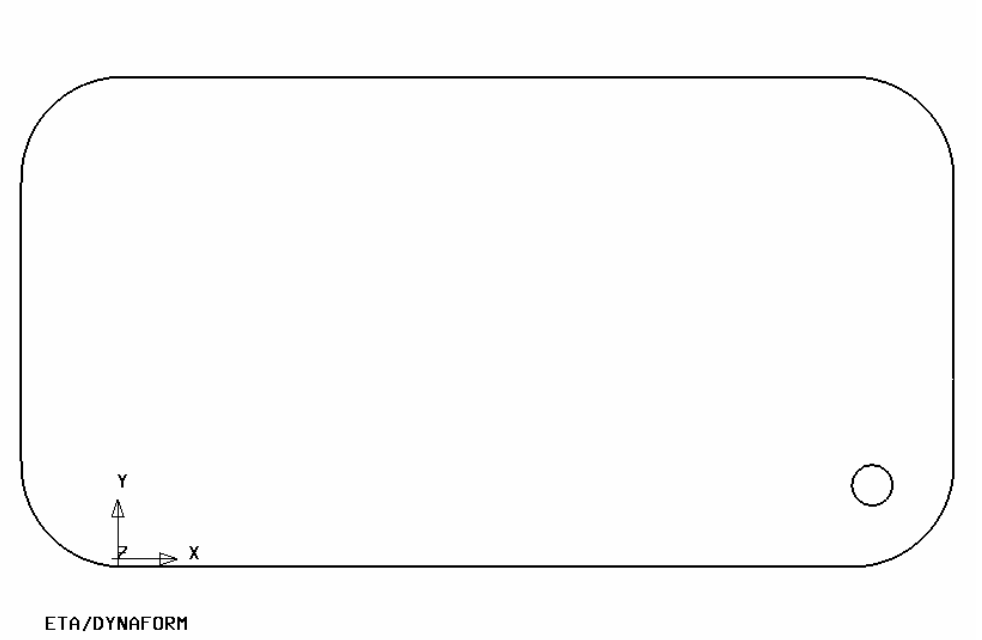


Figure 4.21: Illustration of boundary line

3. Employ the other check functions to check and delete the too small and overlap elements.
4. Click “Clear” in the toolbar to clear the highlighted boundaries. See figure 4.22.

Figure 4.22: Toolbar

- AUTO PLATE NORMAL 

1. Select “**Model Check Auto Plate Normal”** to display a new dialogue window.
2. There are two options in the dialogue window: check all the active parts and only check the cursor pick part. In default, all the active parts will be checked, so user may arbitrarily select one element to adjust the normal direction consistent for all the active parts. Otherwise, user choose the second option, then arbitrarily select one element in the part to be checked to adjust the normal direction consistent for that part. In this example, user may select one element in the medal arbitrarily.
3. A arrow representing the normal direction of the selected normal is shown in the screen, and the popup window prompts “**Is normal direction acceptable?**”, i.e. ask if the user accepts the normal direction. See figure 4.23.

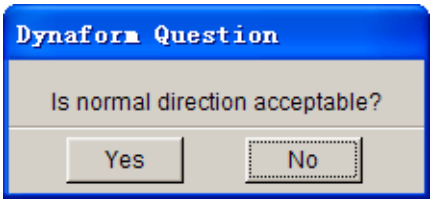


Figure 4.23: Dynaform Question dialog box

4. Click “**YES”,** the normal direction of all the parts are adjusted consistent with the displayed direction. Click “**NO”,** the normal direction of all the parts are opposite to the displayed direction. In this example, select “YES”.
5. Make sure the normal direction of all the parts is consistent for all the parts, then save the database.

V. MSTEP Module and Parameter Set

1. Select menu **BSE→MSTEP** to open the MSTEP solution module. See figure 4.24.

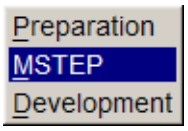


Figure 4.24: BSE menu

2. The following figure is the Schematic View of MSTEP module. The module operation is simple, once assign the corresponding part to tool and select the solution mode; the simulation operation can be performed. See figure 4.25.

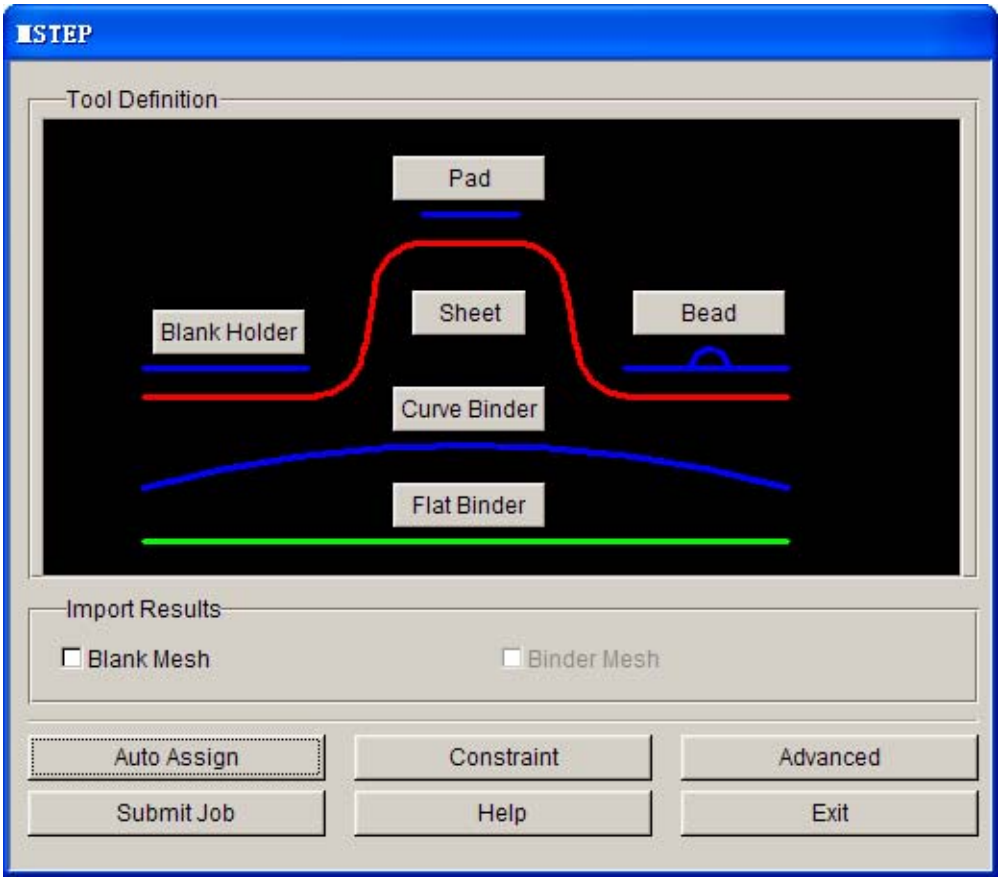


Figure 4.25: MSTEP GUI

3. define tool

   1) Select “Sheet” from the MSTEP dialogue window, then pop up the Define Blanks dialogue window; select “Add” to add the corresponding part to the sheet-metal tool. See figure 4.26.

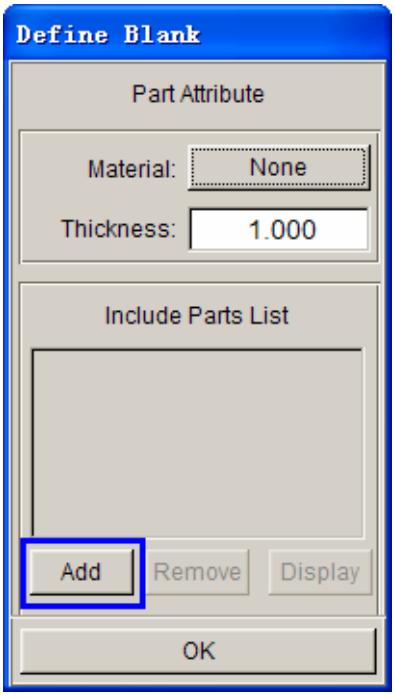


Figure 4.26: Define Blank dialog box

2) In the pop-up Select Part dialogue window, select the part of “BLANK1” as the part corresponding the sheet-metal tool, at the moment all the elements in the selected part are highlighted, as shown in following figures 4.27~4.28.

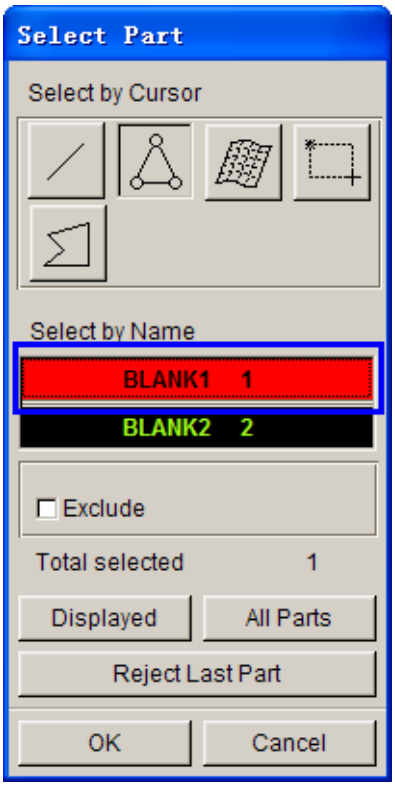


Figure 4.27: Select Part dialog box

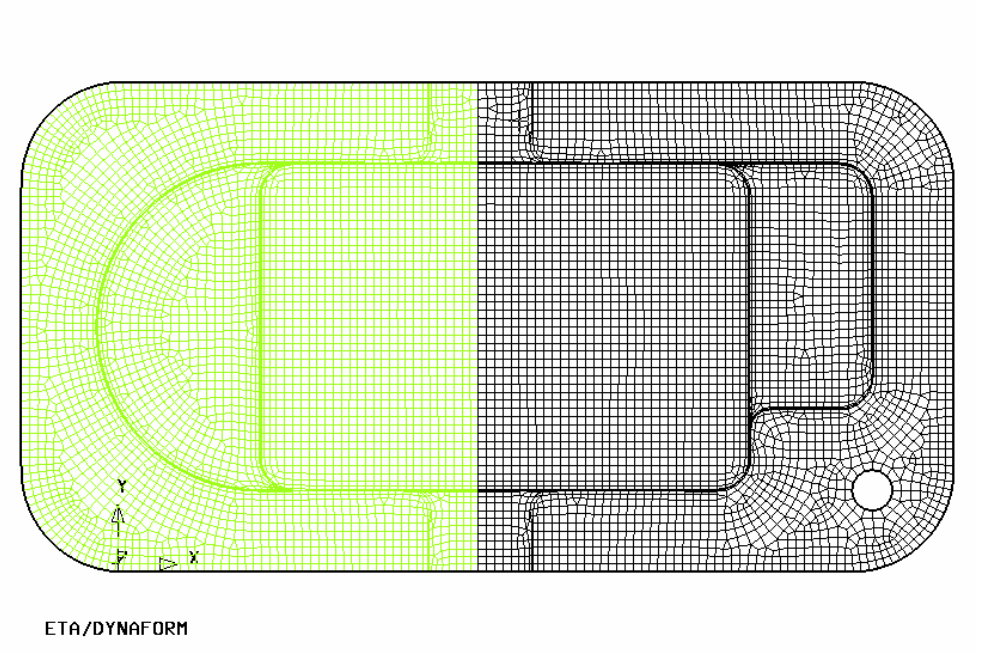


Figure 4.28: Illustration of Selected part

3) Click “OK” to return to the Define Blank dialogue window, user can find the selected target part has been added to the Include Parts List. See figure 4.29.

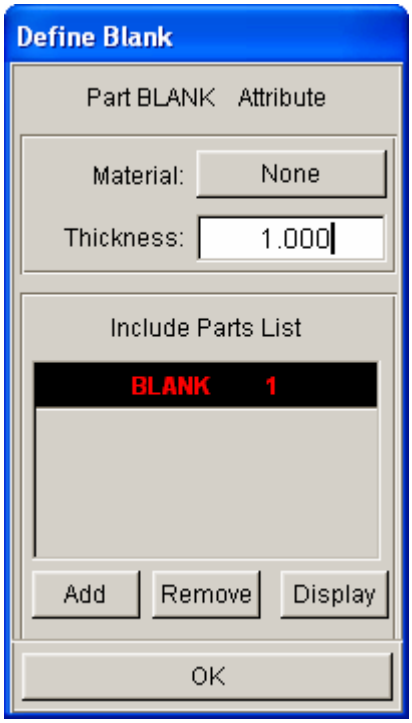
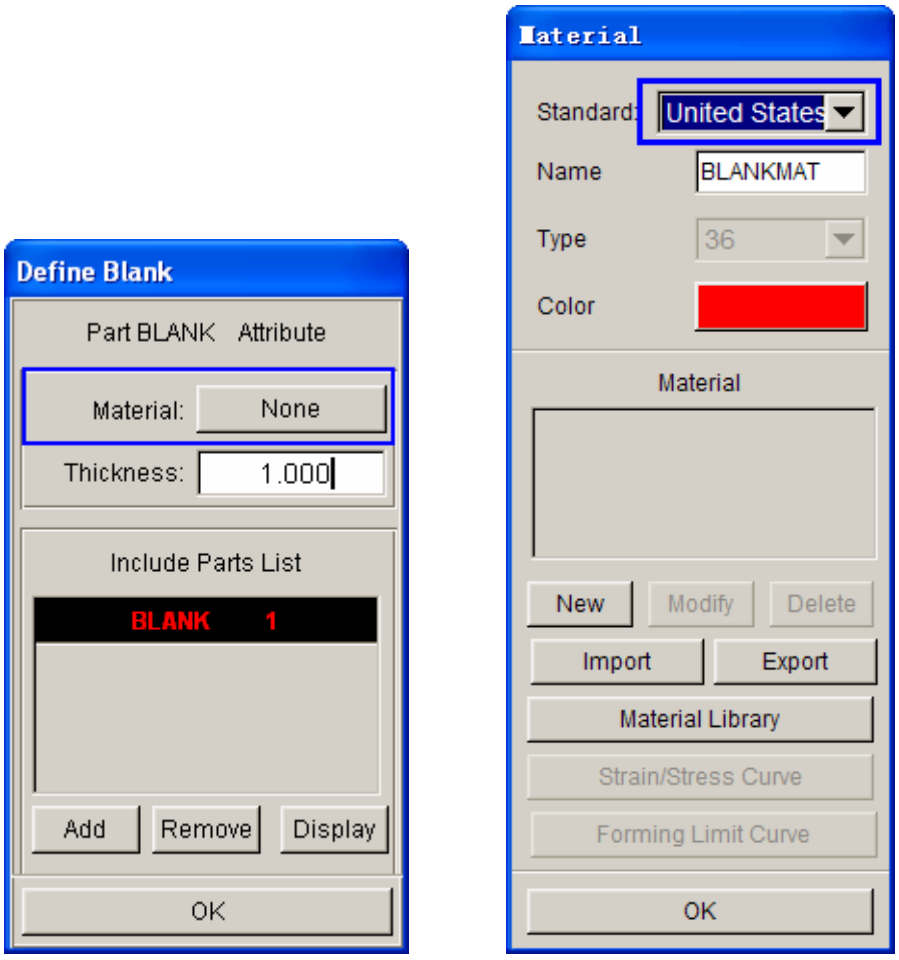


Figure 4.29: Define Blank dialog box

4) Click the BLANK1 from the Include Parts List, the selected part’s color is changed into black; then select “None” behind Material, popping up the Material dialogue window, at the time all the elements in the selected part are highlighted. See figure 4.30.



a) b)

Figure 4.30: Define Blank dialog box

5) Change the Material Standard to **UNITED STATES**.

6) Select the Material Library button from the Material dialogue window, then pop up the dialogue window of Dynaform Material Library, from which select the material of DQSK type 36 as the material type of BLANK part. See figure 4.31.

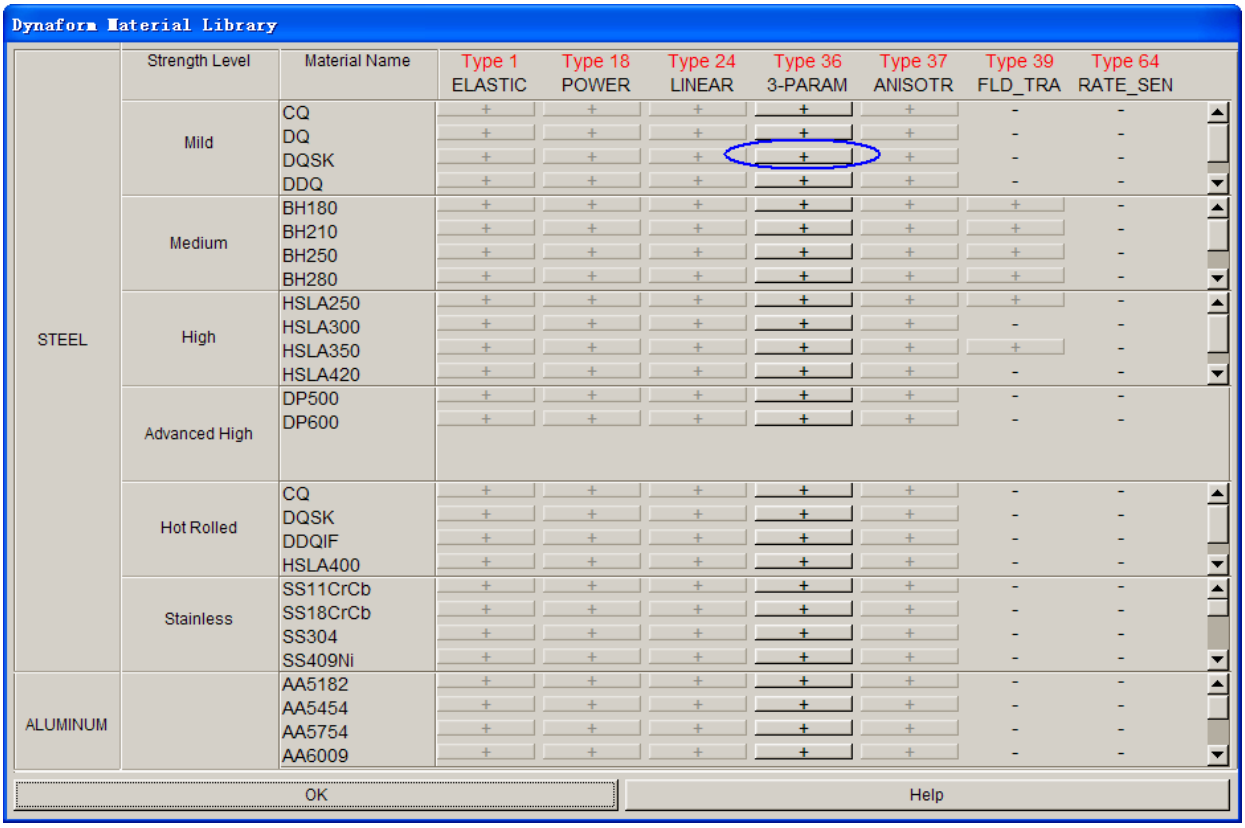


Figure 4.31: Material Library window

7) Click “OK” to return to the Material dialogue window, then user may find the selected material type has already been added to Material List. See figure 4.32.

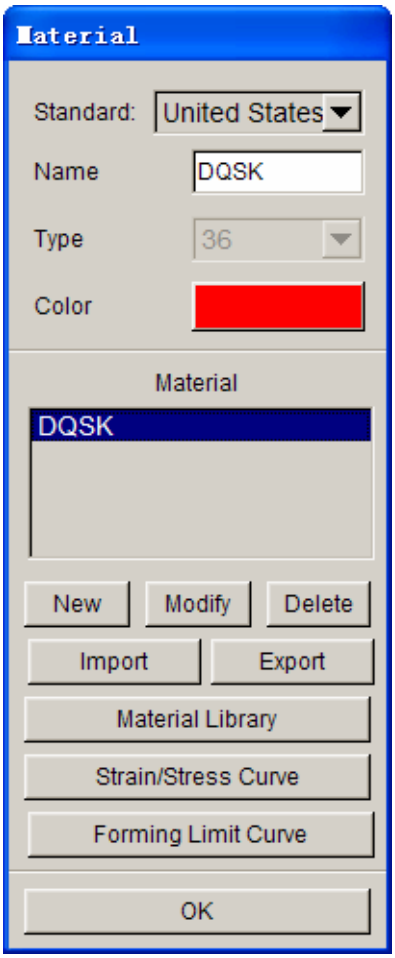


Figure 4.32: Define material dialog box

8) Click “OK” to return to the Define Blank dialogue window，then user may find the None button behind Material has been changed into the selected material type name of DQSK, which indicates the part has been assigned a kind of material. See figure4.33.

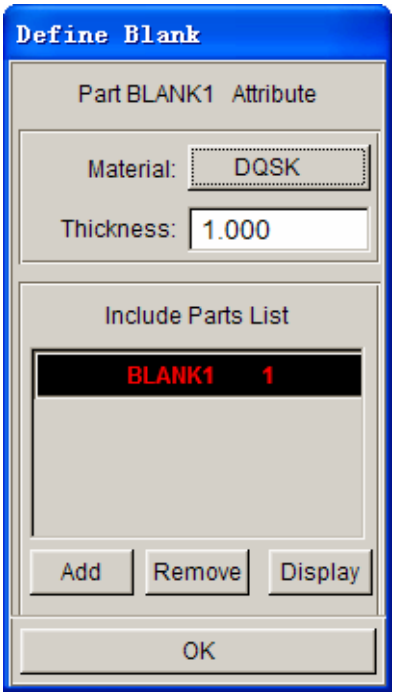


Figure 4.33: Define Blank dialog box

9) In this example, the thickness of the first sheet metal is 1.2 mm, so enter 1.2 in the input box behind Thickness. See figure 4.34.

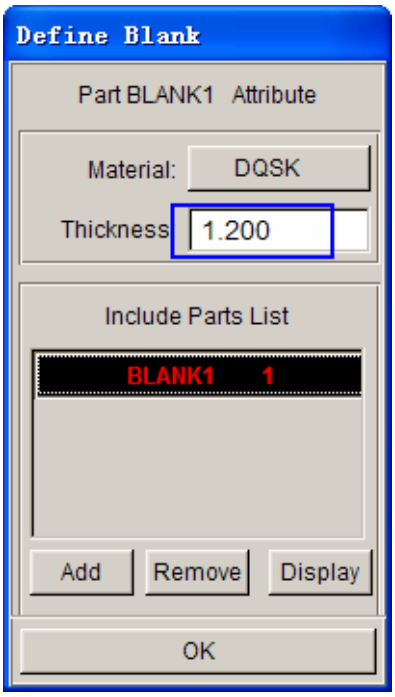


Figure 4.34: Define Blank dialog box

10) Repeat the steps above, assign the part of BLANK2 to the part corresponding to the sheet metal tool, and define the material type and property of that part. Wherein the material type is AA5182 type 36, the material property is blank2, and the thickness is 0.7 mm. See figure 4.35.

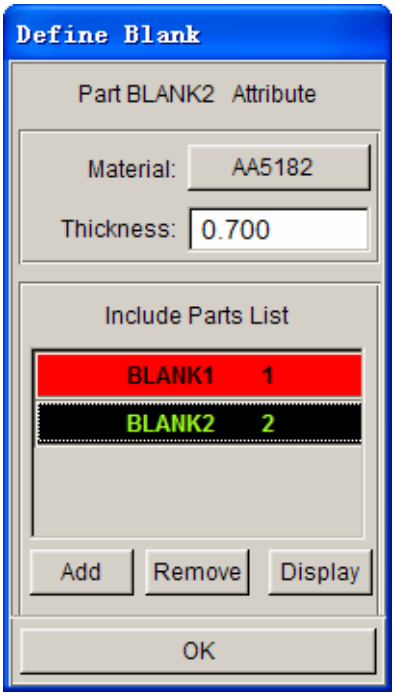


Figure 4.35: Define Blank dialog box

11) Click “OK” to return to the MSTEP dialogue window, user can find the outline representing the sheet metal in the dialogue window has changed from red into green, which indicates the define of sheet-metal tool has been completed. See figure 4.36.

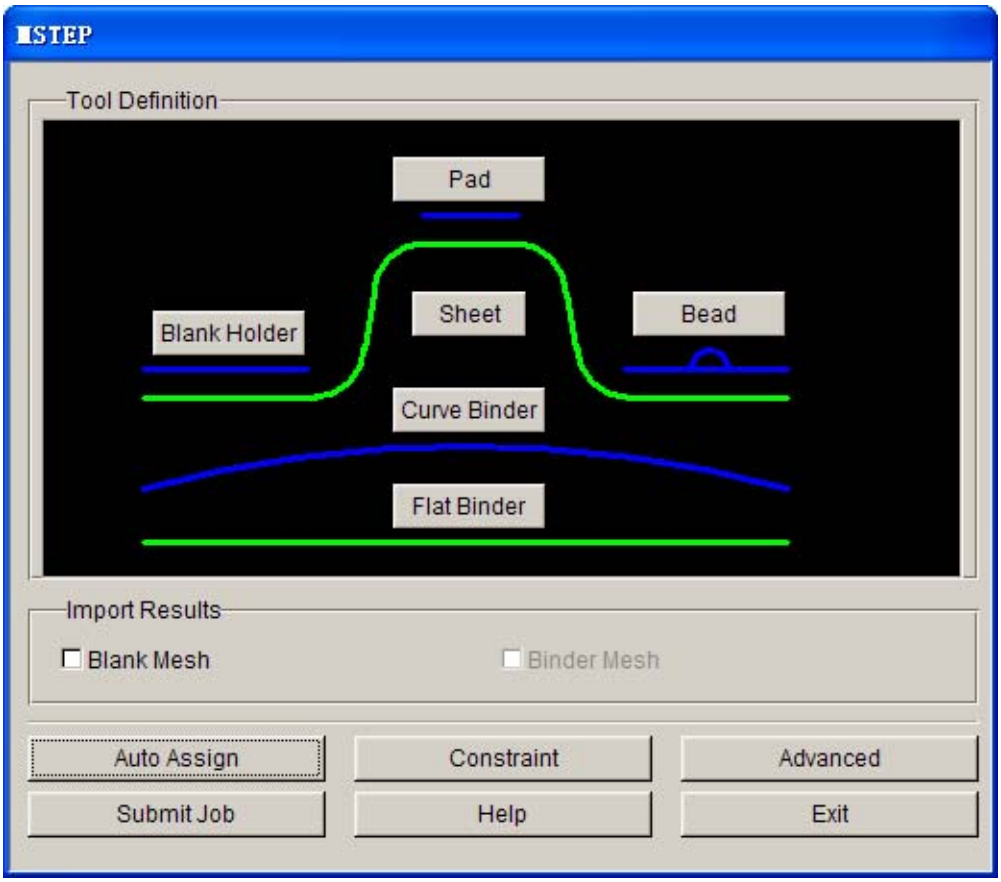


Figure 4.36: MSTEP of sheet definition

4. define simulation parameter

G dialogue

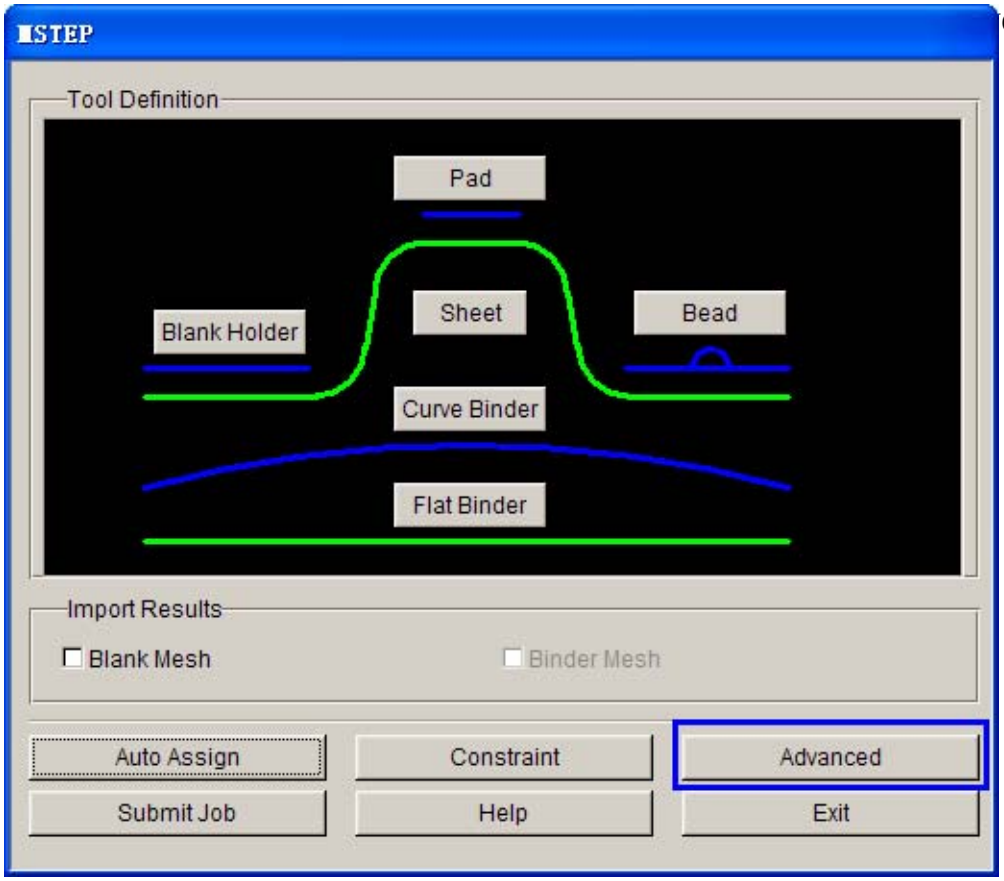


Figure 4.37: MSTEP GUI

2) The MSTEP module provides two solution modes: Accurate and Fast. Wherein the Accurate mode considers the impacts of the binder hold force, pad hold force and draw bead, and the impacts of material parameters and plasticity deformation factors, which is perfectly consist with the real formability process and the result is most accurate, so it is adapted to analyze the product formability in the conceptual design, obtain the accurate sheet blank outline, collate the process planning, and estimate the impact of the process parameters on the product forming process. The Fast mode ignores the impacts of above parameters in solution, so it's unnecessary to set the related tools, such as Blank Holder, Pad, Bead etc., it's suit for sketchily estimating the product deform and the sheet unfold outline.

In this example, the offset of the tailor welded line and original formability should be considered, so select the Accurate mode, and set the binder hold force at 200,000 Newton based on fact and accept other default values, the result is shown in the following figure 4.38.

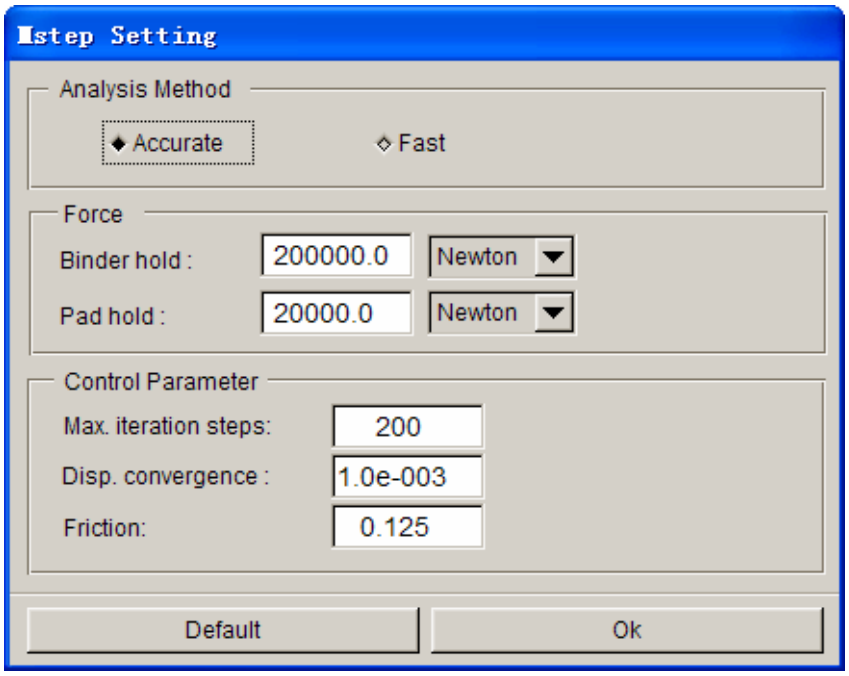


Figure 4.38: MSTEP Setting dialog box

3) Click “OK”, return to the MSTEP dialogue window.

5. start up the MSTEP solver

Now, in this example, all the solution related parameters have been defined, and then select the Submit Job button in the MSTEP dialogue window to start up the MSTEP solver. The solution operation is performed. See figure 4.39.

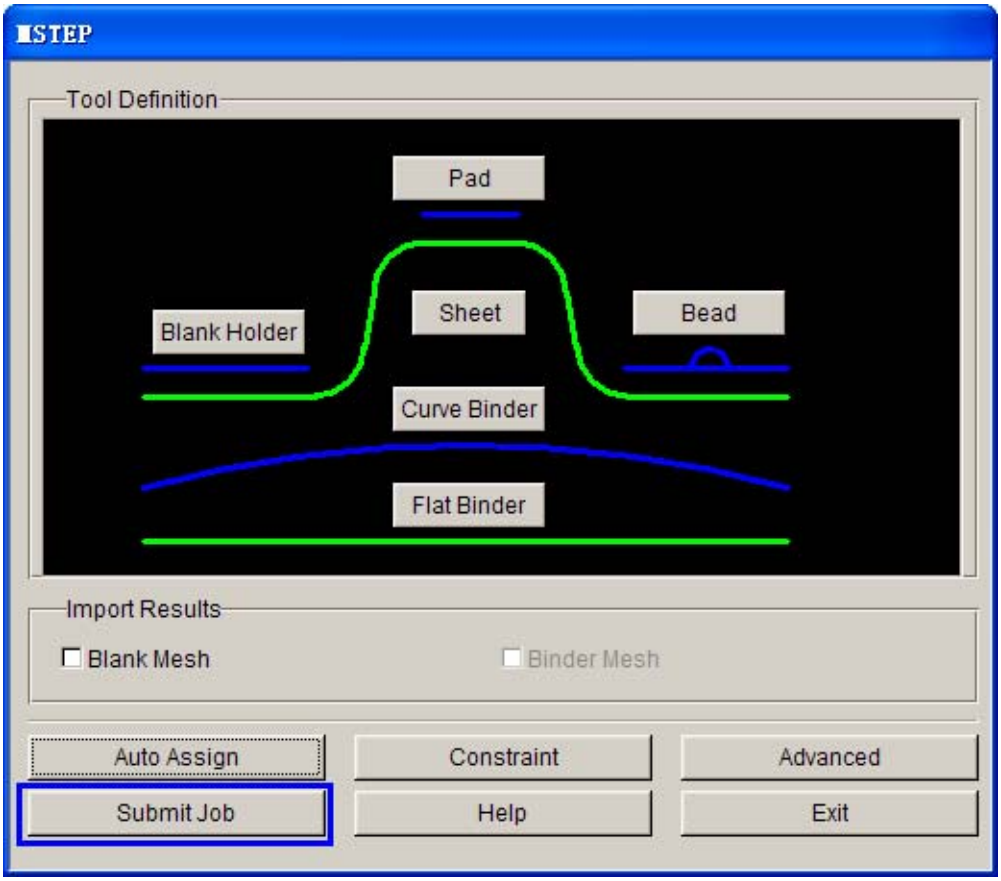


Figure 4.39: MSETP GUI

VI. Start up Post-Process and Analyze Simulation Result

After the MSTEP evaluation is finished, DYNAFORM will automatically read in the resulting sheet blank unfold outline, and add it to a new part, the blue closed line shown in the following figure is the part's sheet blank unfold outline. In this example, the inner contour will be badly deformed after it unfold to initial plane sheet blank, so it's prefect that the punch is performed after the formability，in order to obtain the desire result. See figure 4.40.

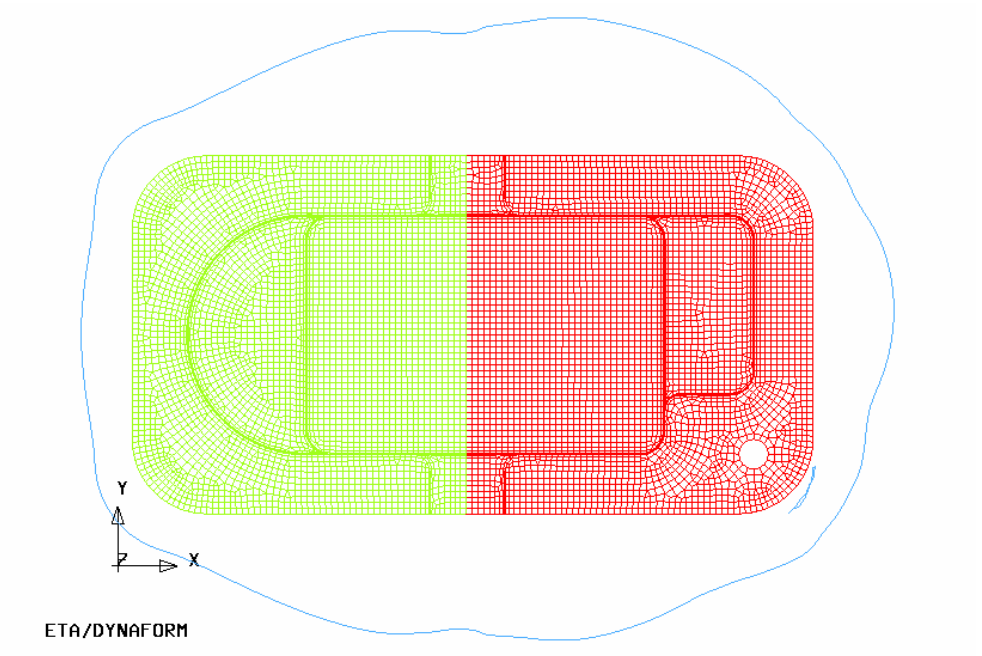


Figure 4.40: Illustration of model result

In order to observe the full information about the result, User may start up the Post-process program to analysis the result. In the result from MSTEP module, the emphasis should be laid on observing the forming limit diagram (FLD) and the thickness distribution diagram to check the flow and formability defects of sheet blank.

1. Select the menu of PostProcess to start up the ETA/Post-Processor.

2. Select menu **File→Open** as shown figure 4.41 **or** click the OPEN icon 

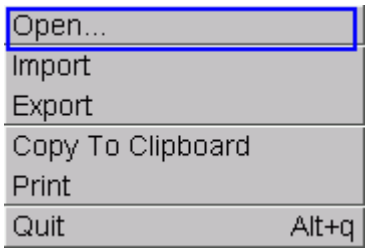


Figure 4.41: Open file menu

3. Pop up the dialogue window to open file, select the file of dynain.mstep, and click “Open”, then the result file will be read in, which is shown in the following figure 4.42.

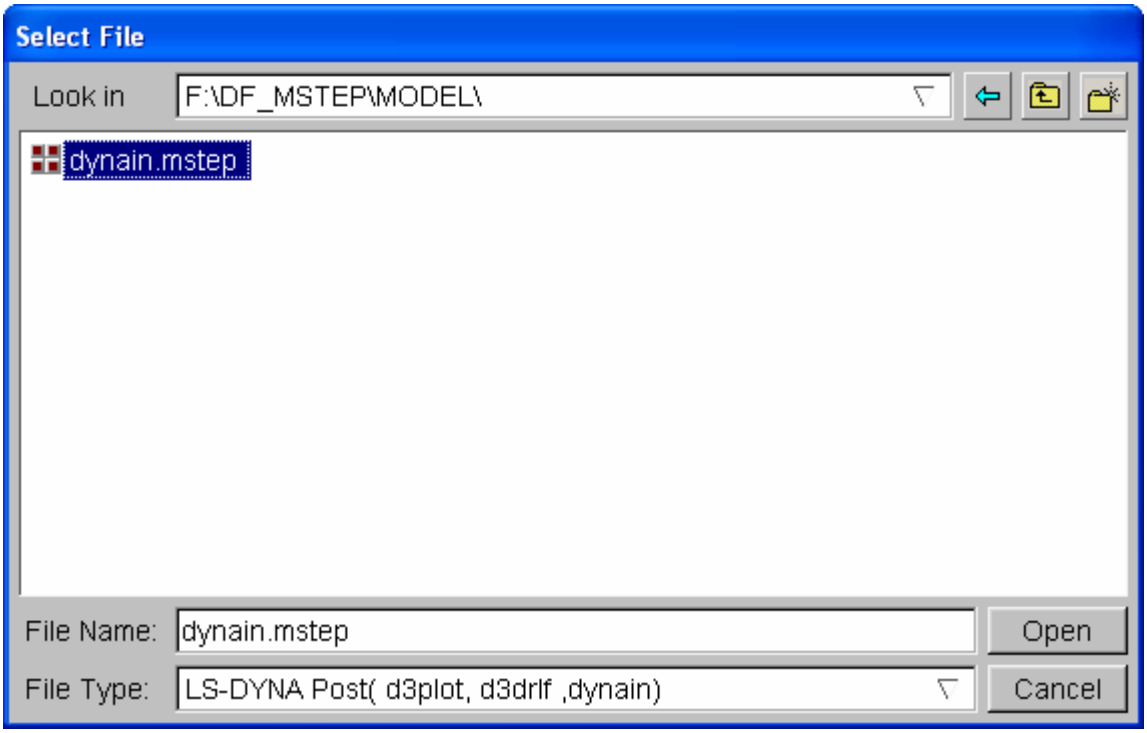


Figure 4.42: Open file window

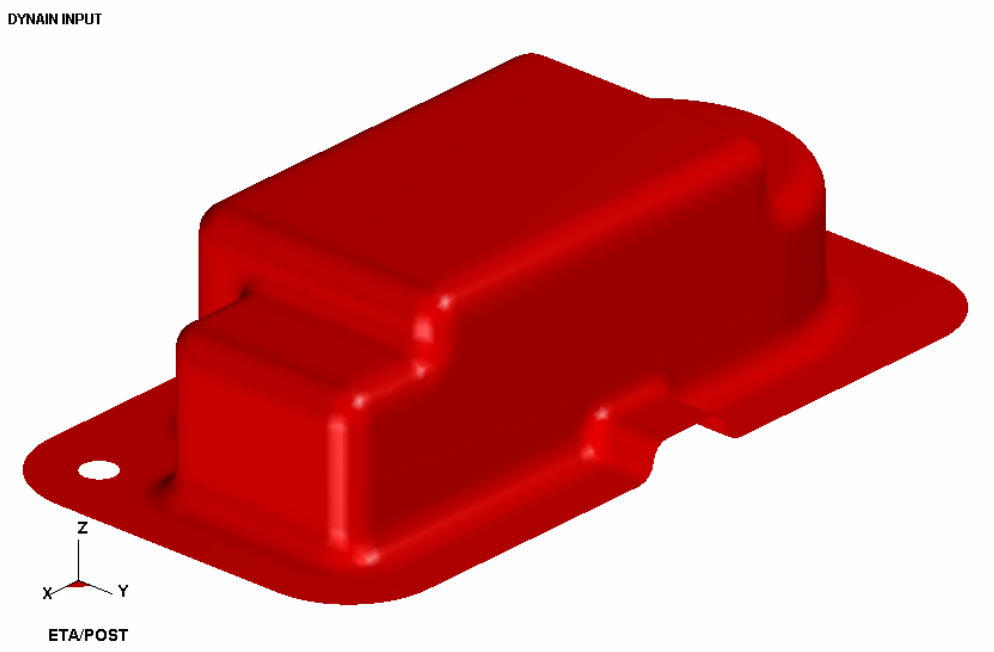


Figure 4.43: Illustration of dynain.mstep

4. forming limit diagram as shown figure 4.44.

Figure 4.44: Toolbar

1) Select “FLD” in the result operation menu.

2) Select “Middle” in the pull-down menu of **Current Component.** See figure 4.45.

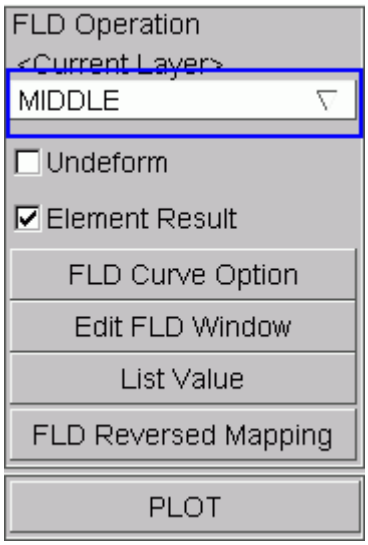


Figure 4.45: FLD dialog box

3) Click “**FLD Curve Option”** to set FLD parameters (n，t，r etc.).

Note：*The eta/Post don’t support that the two sheet metals made from different material are simultaneity displayed in the corresponding diagrams of FLD. So user must define the FLD parameters in twice.*

4) Select “**Edit FLD Window”** to define the position of plotting FLD.

5) Click “PLOT” to plot the forming limit diagram of the finished component, as shown in the following figure 4.46.

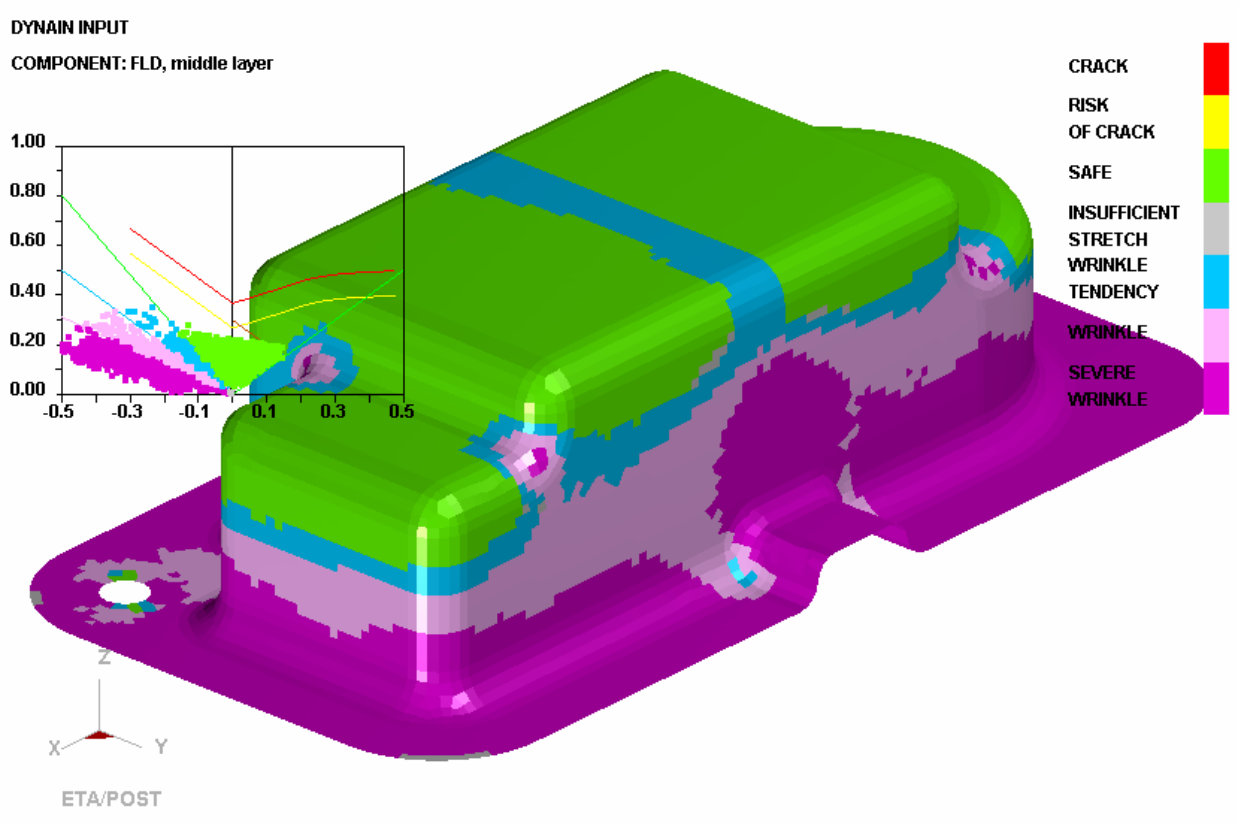


Figure 4.46: Illustration of FLD result

In this example, some areas are displayed in blue and pink when the FLD is shown, which indicates that the components in these areas have a tendency to wrinkle, and the process planning is not meet the formability require. So the user may return to the pre-processor to set the larger binder hold force or set the bead, more information may be obtain from the Dynaform5.2 Mstep formability analysis and the process parameters optimize training manuals.

5. thickness / thinning animation as the following figure 4.47.

Figure 4.47: Toolbar

1) Select "**Thickness"** from the result operation menu.

2) Select THICKNESS (absolute value) or THINNING (relative thinning rate) arbitrarily from the drop down menu of **Current Component**. See figure 4.48.

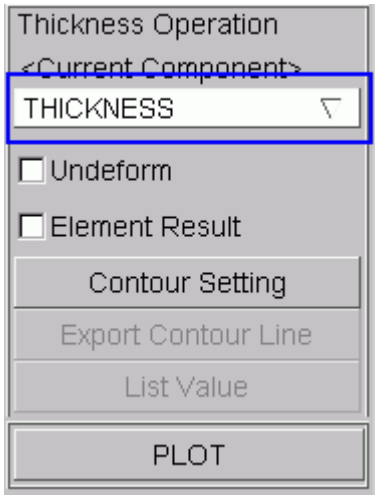


Figure 4.48: Thickness dialog box

3) Click “PLOT” to plot the thickness contour diagram, which is shown in the following figure 4.49.

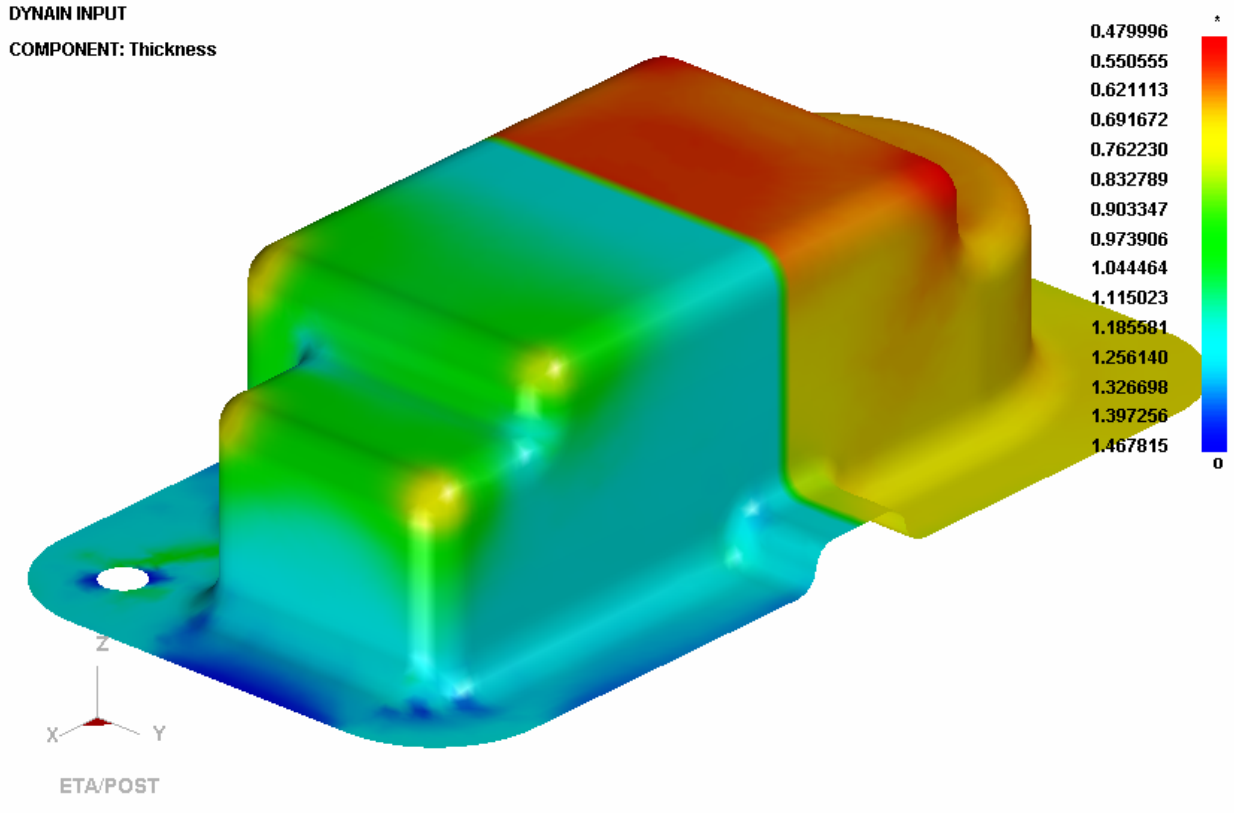


Figure 4.49: Illustration of Thickness result

6. material flow path

1) Select menu **File→Import** to open the Import dialogue window. See figure: 4.50.

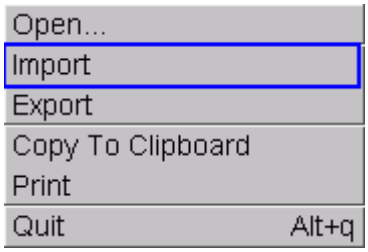


Figure 4.50: Import file menu

2) Select the file of MSTEP_model3_mstep.lin, click “Open”, and the sheet blank outline is read in, as shown in the following figures 4.51~4.52.

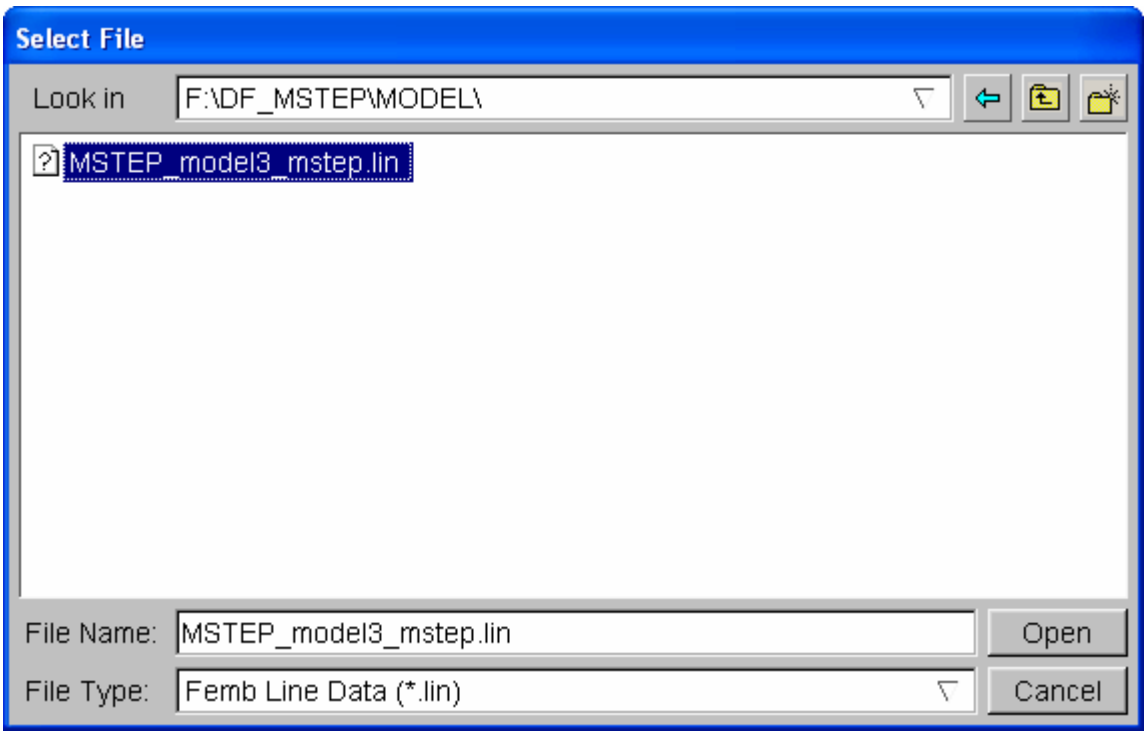


Figure 4.51: Open file window

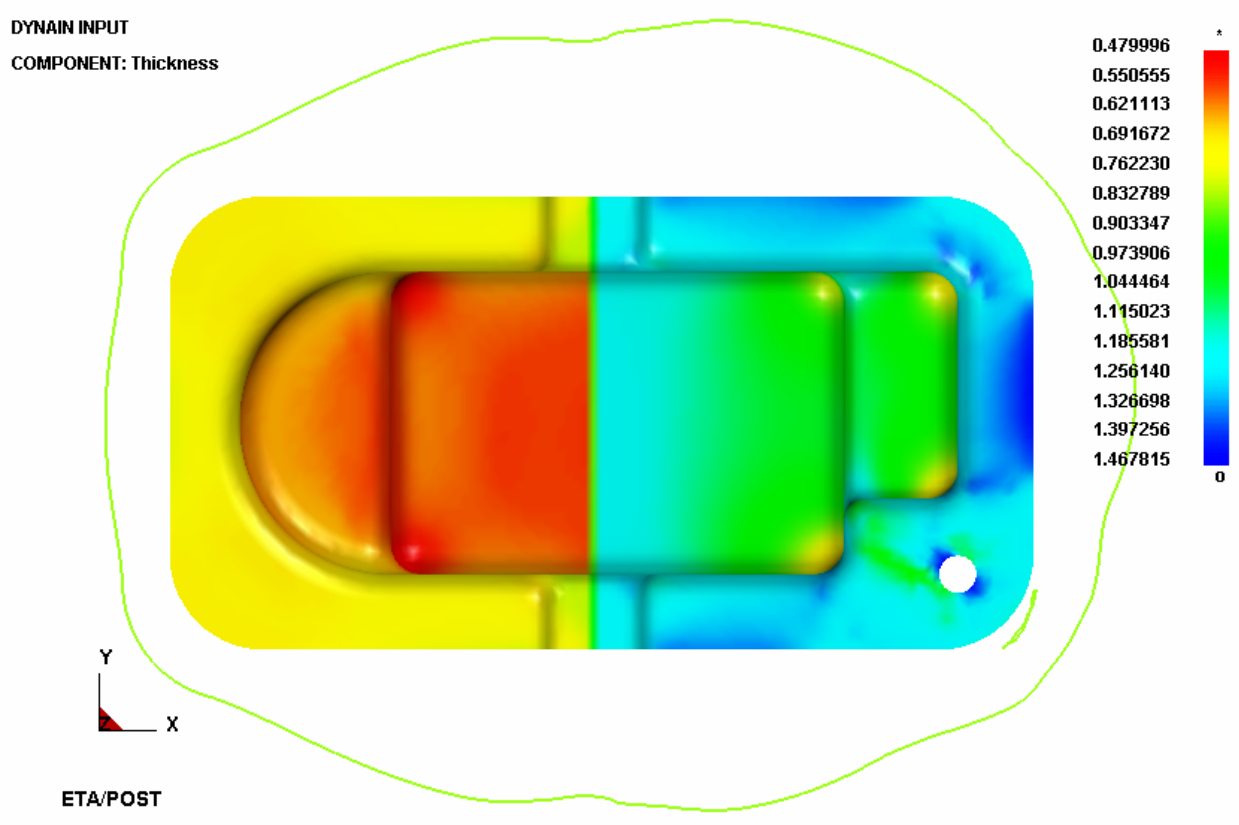


Figure 4.52: Illustration of Model result

The result:

With regard to the formability simulation of the tailor welded blank, the basic design idea is that the different material correspond the different parts, for example, the two parts are created in this example, then obtain the exact position of the initial welded line and the change tendency of the welded line in formability process according to the displacement change of the nodes in the common boundary of the components.

In modeling, the attention should be laid to that the corresponding parts are based on the material and meshed, but all the meshes are connected each other; the surface can't mesh separately, otherwise error will occur. The following figure 4.53 shows the simulation result and the blank unfold outlines, which are obtained by separately mesh the two parts corresponding to the sheet metal tool.

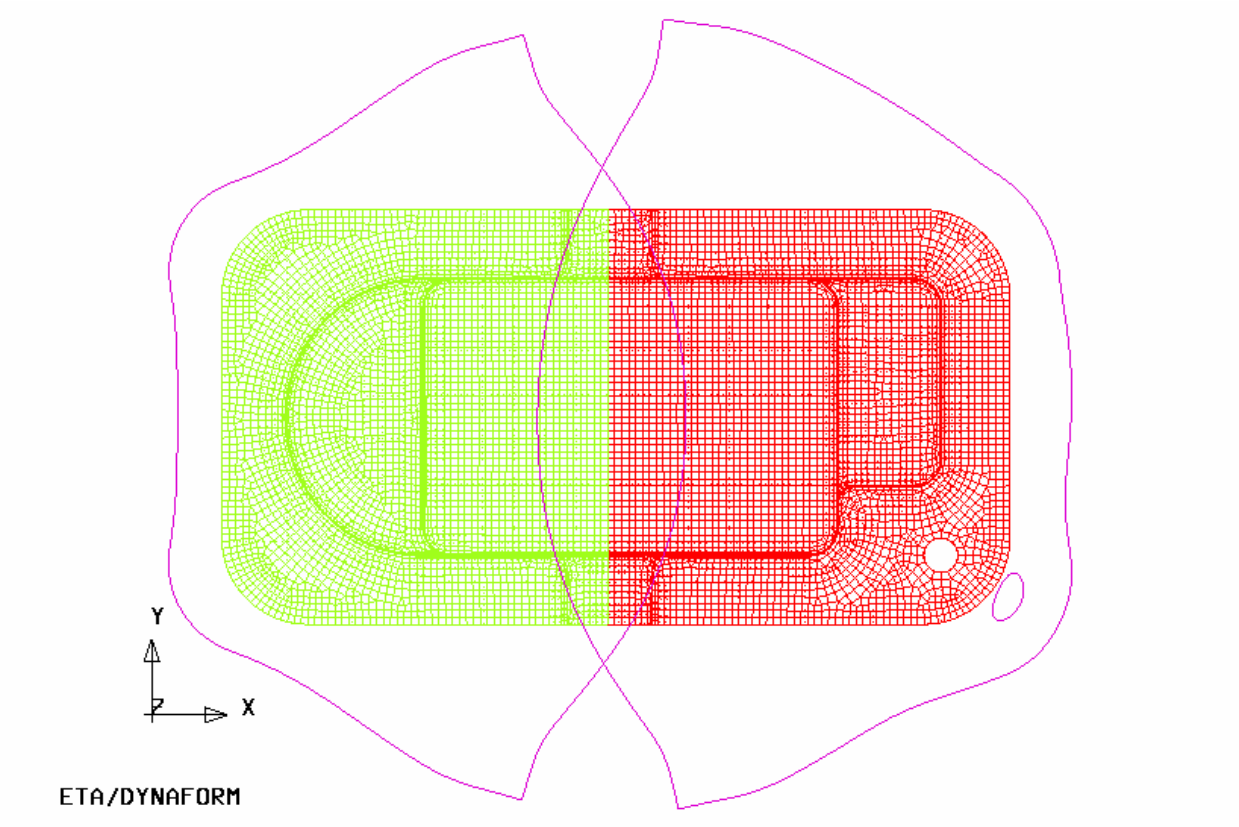


Figure 4.53: Illustration of blank unfold outlines