Thermal Forming & Cooling

Thermal Forming & Cooling

Engineering Technology Associates, Inc.

DYNAFORM 5.8 Training Tutorial

Contents

Note: 1. The input value of the parameters should convert to the unit in the GUI.
2. The unit in the input deck is: MM, TON, SEC, N, K

I. Import the mesh and create a new setup

1. Click the menu File/Import: select the *thermal_case01.dat* file. (Figure 1).
2. Click the menu AutoSetup/Sheet Forming. (Figure 2).
3. Define the new simulation as illustrated in Figure 3.
4. Click ***OK*** to continue.

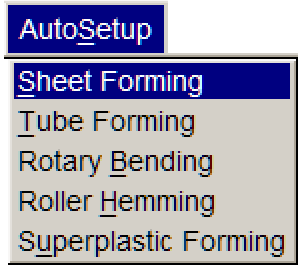

Figure 2

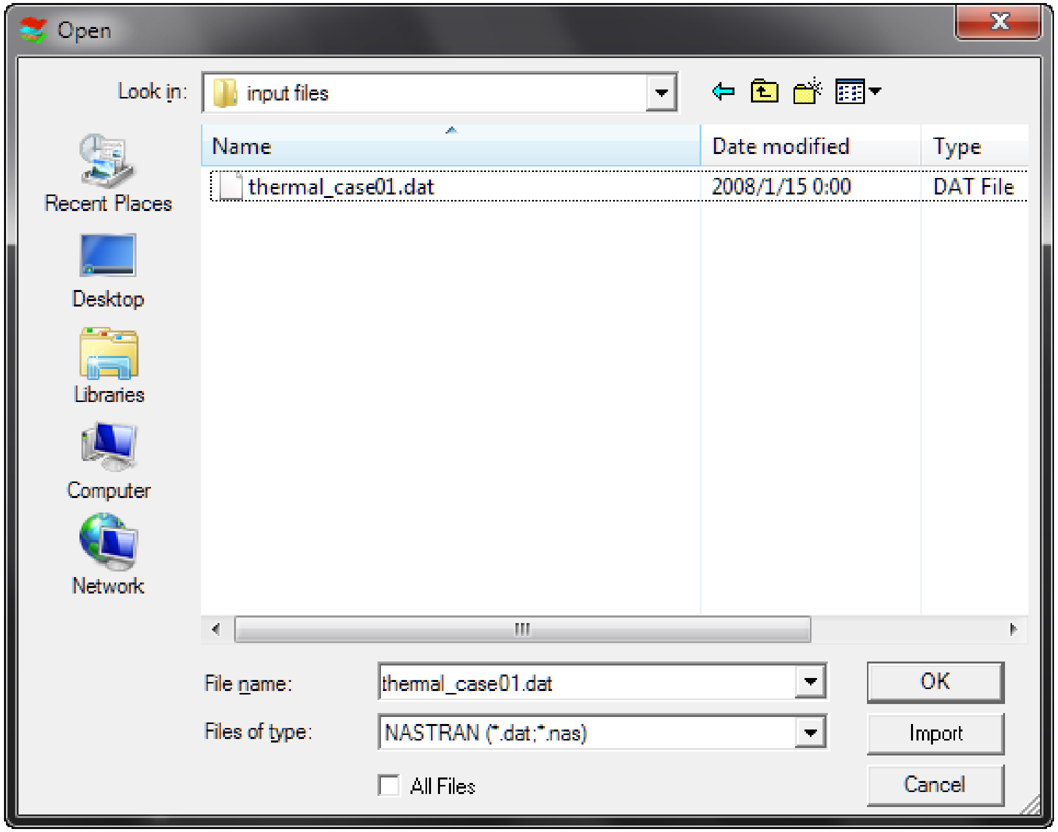

Figure 1

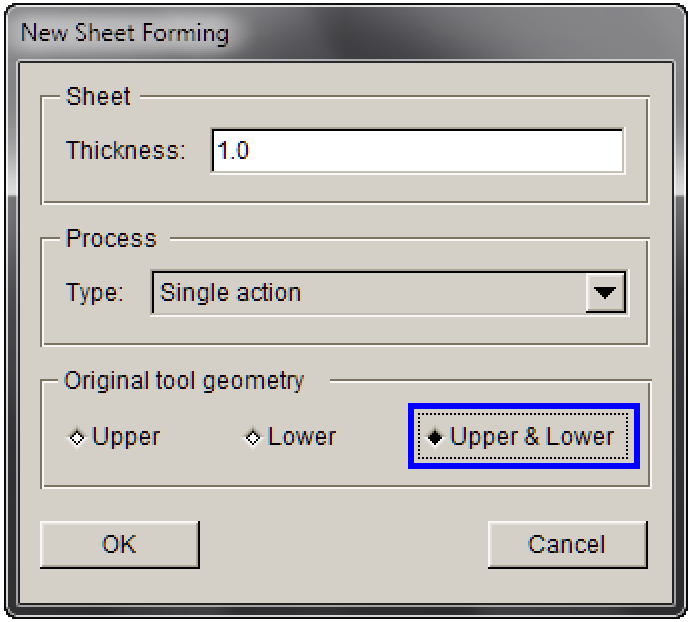

Figure 3

II. Define blank part

1. Keep the General property default.
2. Click ***Blank*** illustrated in Figure 4.
3. Click ***Define Geometry*** button to display the Blank generator dialog box.
4. Click ***Add Part…*** (Figure 5).
5. Select part P0000001.
6. Click ***OK*** to define it as the blank part (Figure 6).
7. Click ***Exit*** button (Figure 5).

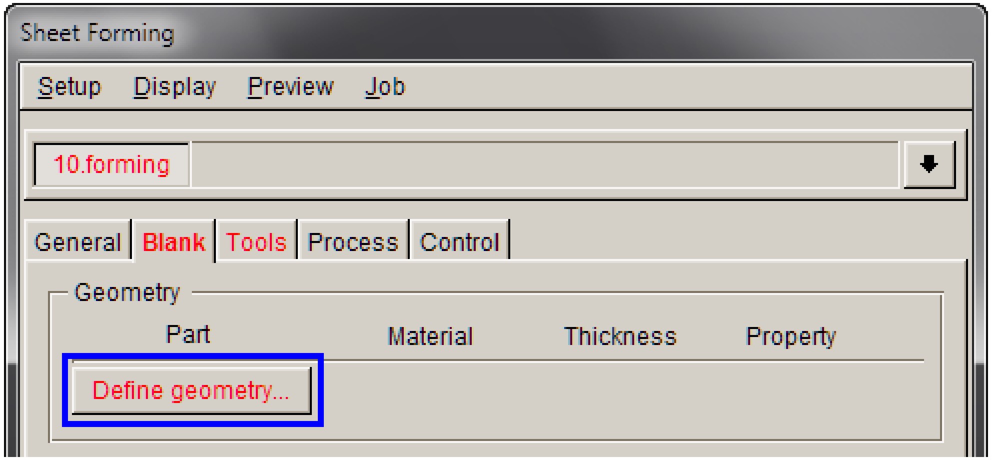

Figure 4

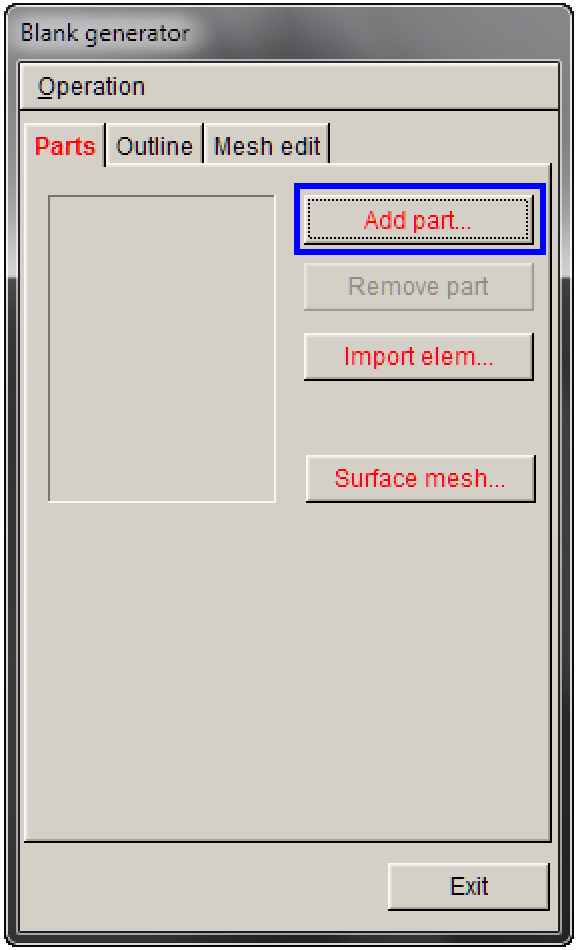

Figure 5

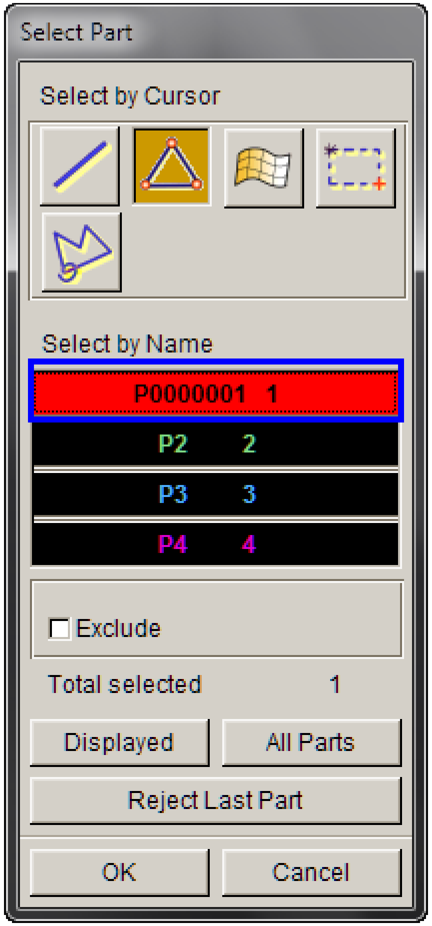

Figure 6

III. Define element formulation

1. Click the button below Property (Figure 7) .
2. Define the element formulation as 16.Fully Integrated (Figure 8).

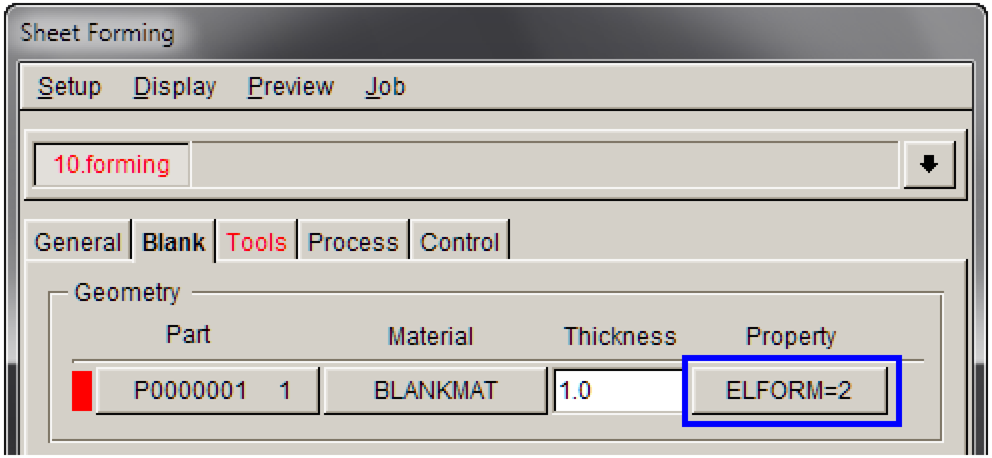

Figure 7

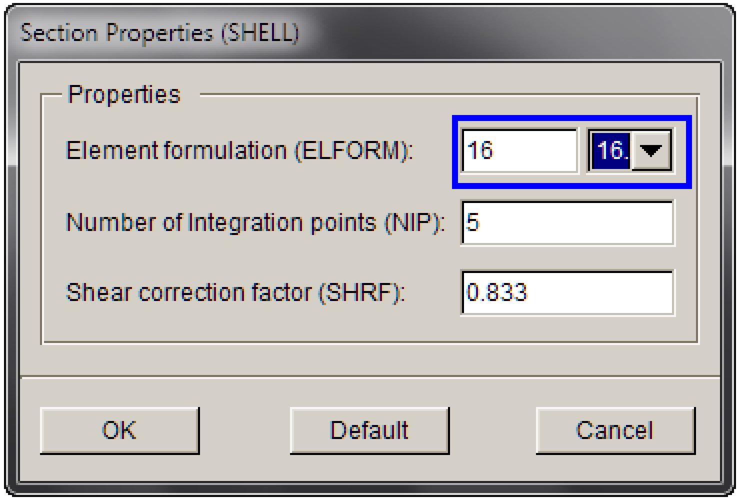

Figure 8

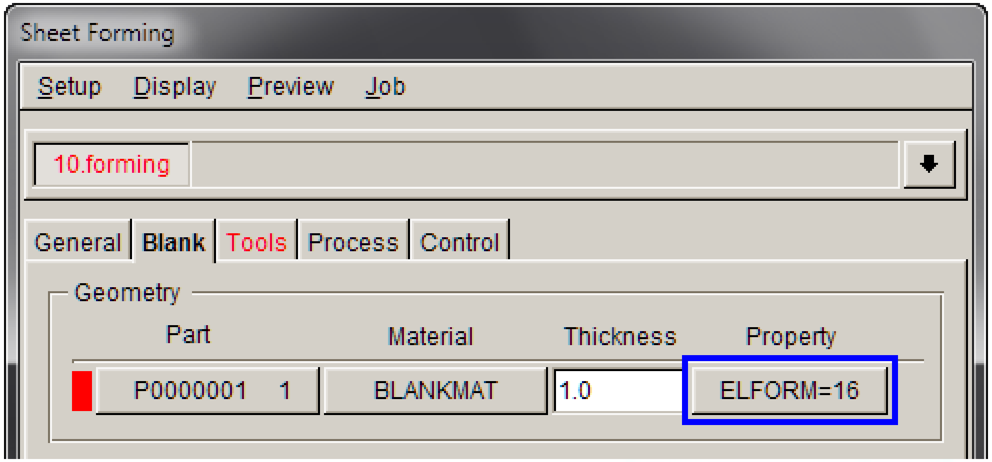

Figure 9

IV. Define blank structural material

1. Click the button below Material to define the structural material model (Figure 10).
2. Select the model 106*MAT_ELASTIC_VISCOPLASTIC_THERMAL (Figure 11).
3. Material parameters dialog box is illustrated in Figure 12.

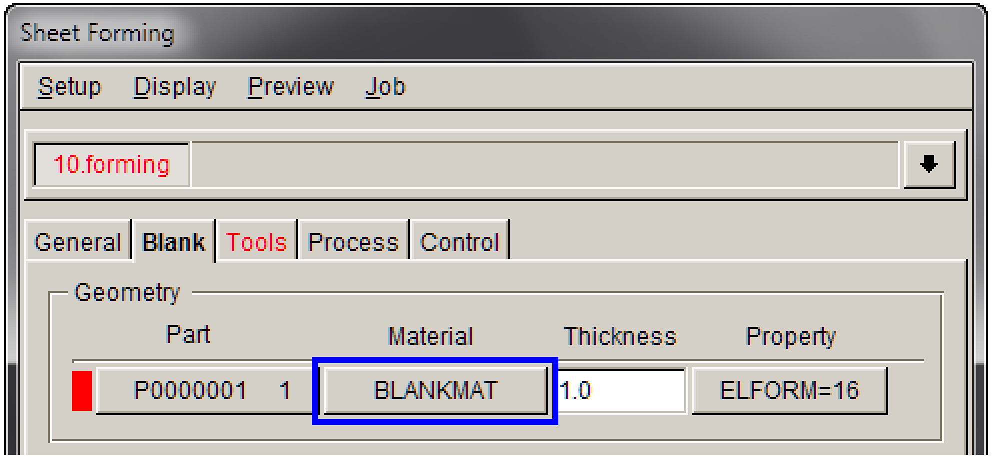

Figure 10

IV. Define blank structural material

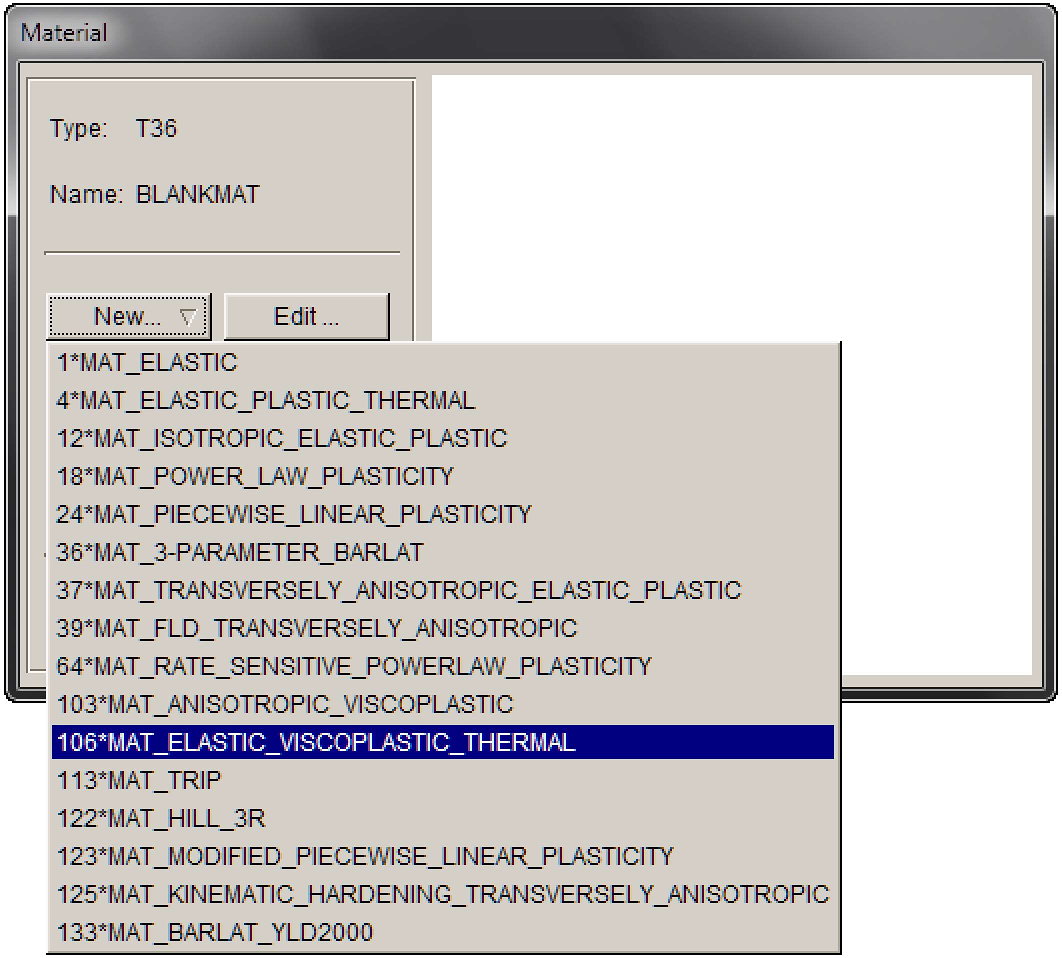

Figure 11

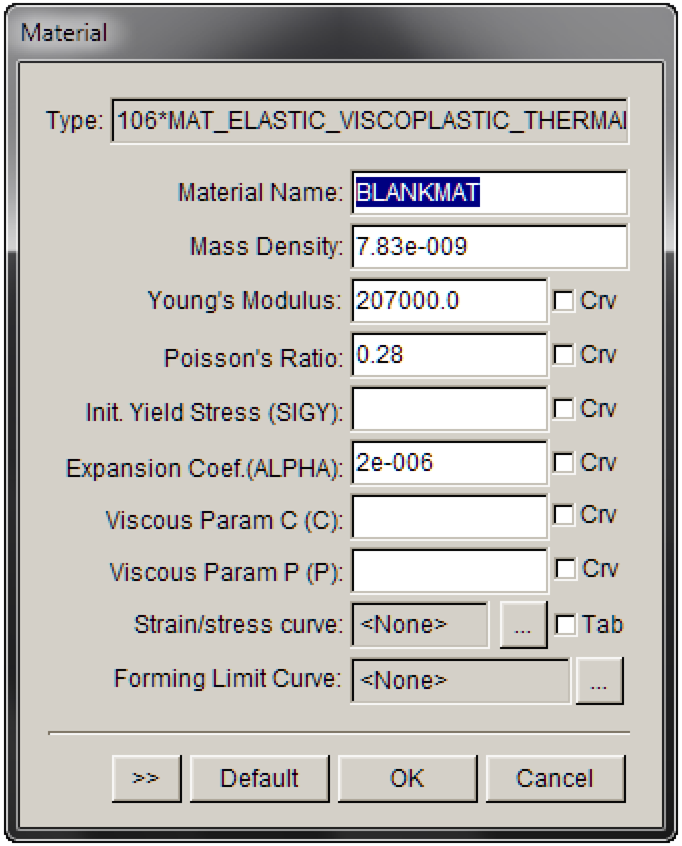

Figure 12

Note: Structural Material Model For Thermal Forming

- *MAT_ELASTIC_PLASTIC_THERMAL (#4)
- *MAT_ORTHOTROPIC_THERMAL (#21)
  - Linearly elastic material
  - Orthotropic temperature dependent coefficients
- *MAT_ELASTIC_VISCOPLASTIC_THERMAL (#106)

  (The #106 is defined in this case)

IV. Define blank structural material

4. Toggle on switch illustrated in Figure 13.
   - Load Curve defining Young's modulus as function of temperature.
   - Load Curve defining Poisson's ratio as function of temperature.
   - Load Curve defining the coefficient of thermal expansion as function of temperature.
   - Load Curve for scaling the viscous material parameter C as a function of temperature.
   - Load Curve for scaling the viscous material parameter P as a function of temperature.
   - Load Curve ID. Once Tab is toggled on, the user needs to define a table for effective stress versus effective plastic strain.

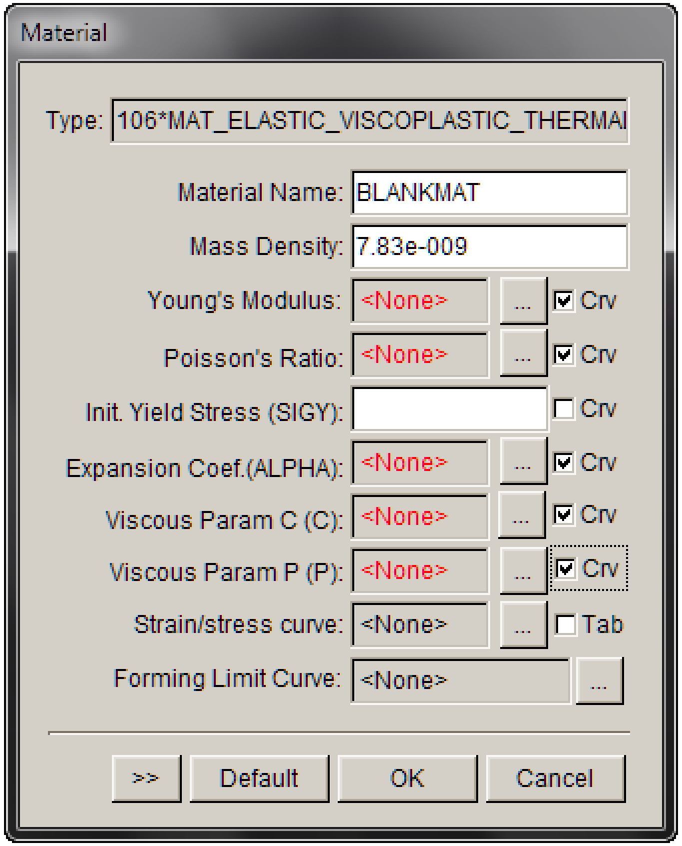

Figure 13

IV. Define blank structural material

5. Click the Young's Modulus button illustrated in Figure 14.
6. The dialog box illustrated in Figure 15 pops up. Click ***Import*** button in Figure 15.
7. Select the file: LCE-Young.cur.
8. Click ***OK***.

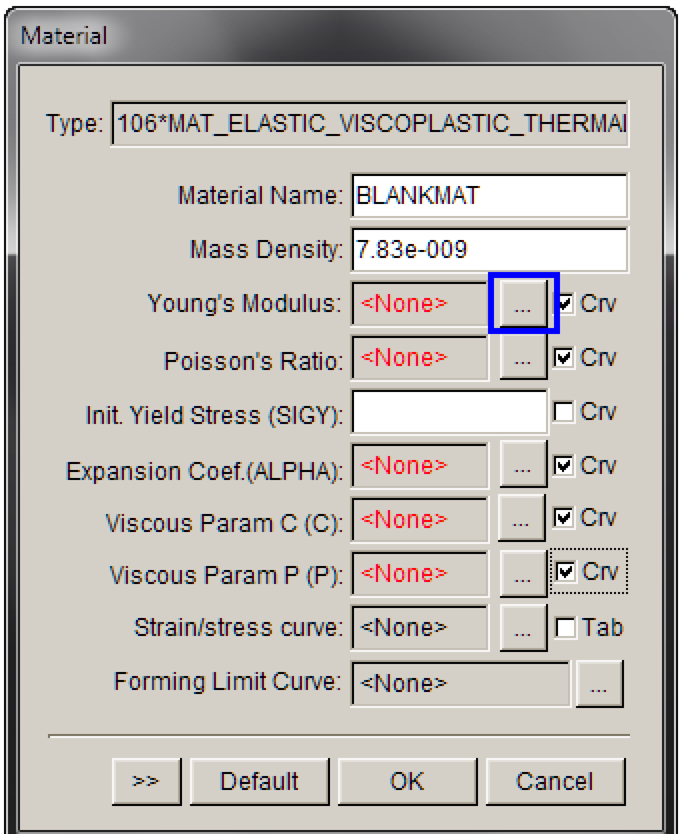

Figure 14

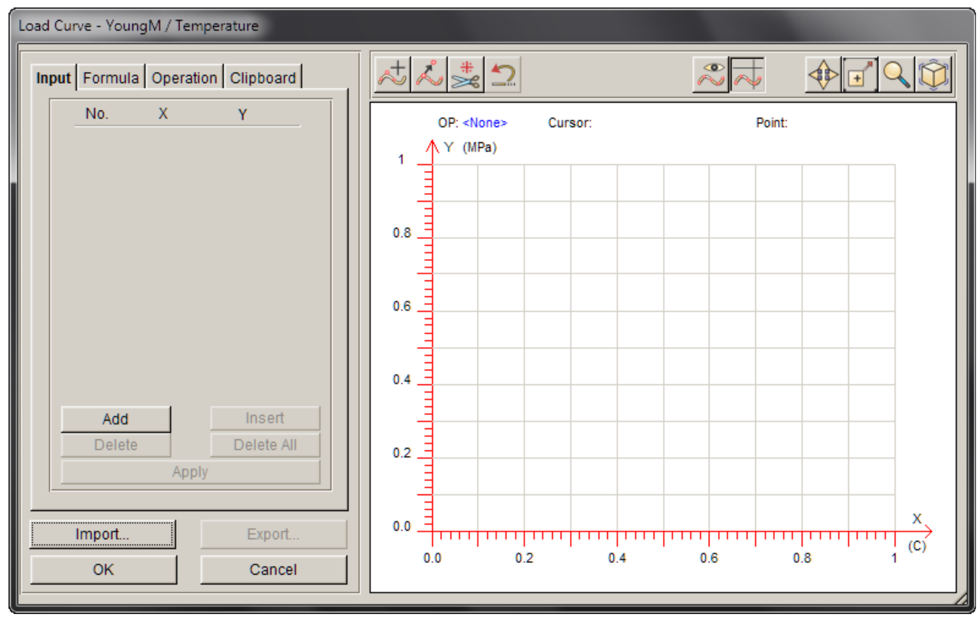

Figure 15

IV. Define blank structural material

9. The imported curve illustrated in Figure 16.
10. Click ***OK*** to accept the curve illustrated in Figure 17.

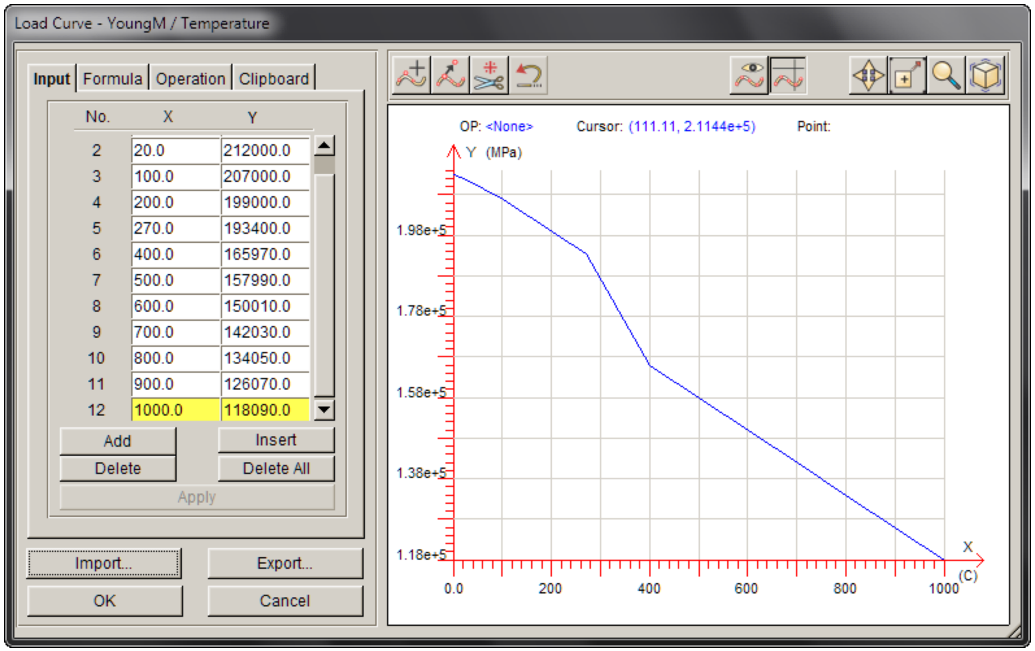

Figure 16

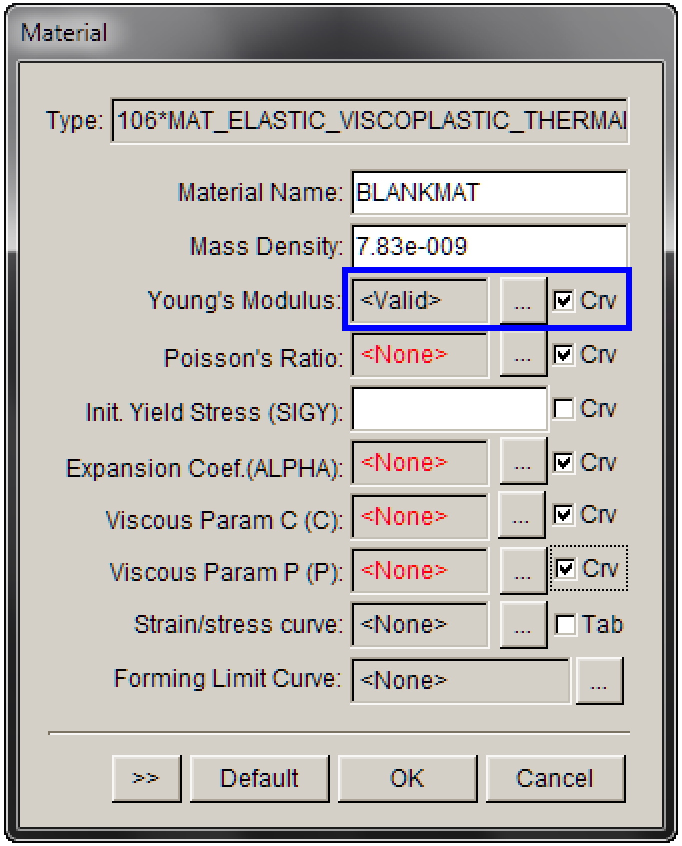

Figure 17

IV. Define blank structural material

11. Refer to the above steps (step 5-step 8) to import the following curves:
    - Select LCPR-Poisson.cur as Poisson’s Ratio;
    - Select LCALPHA-Expansion.cur as Expansion Coef. (ALPHA);
    - Select LCC-C-scaled.cur as the Viscous Param.C;
    - Select LCC-P.cur as the Viscous Param.P.

    The result is shown in Figure 18.

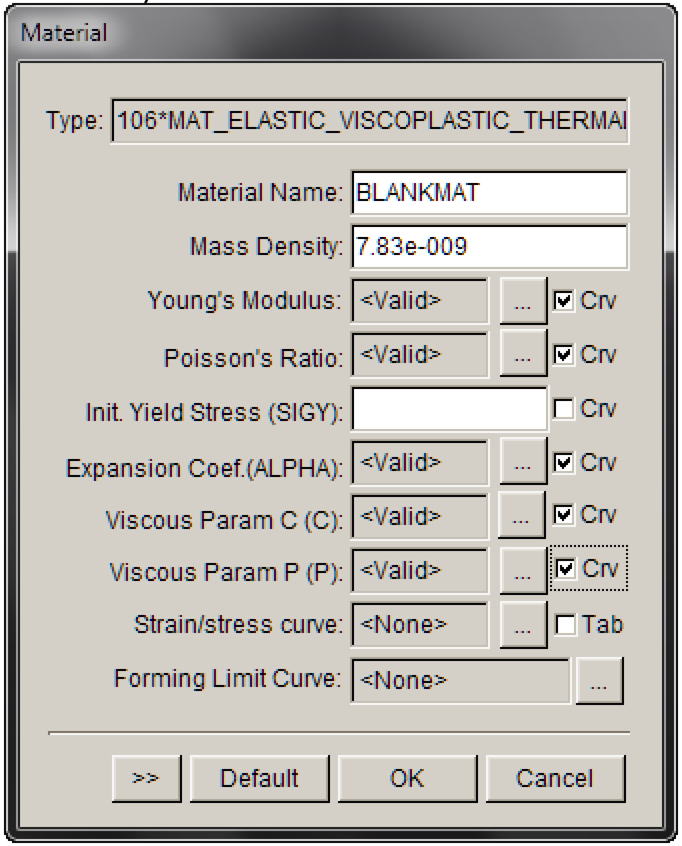

Figure 18

IV. Define blank structural material

1. Click the button left to Tab illustrated in Figure 19.
2. The Table pops up. Click ***Add*** in Figure 20.
3. Click ***None*** in Figure 21.

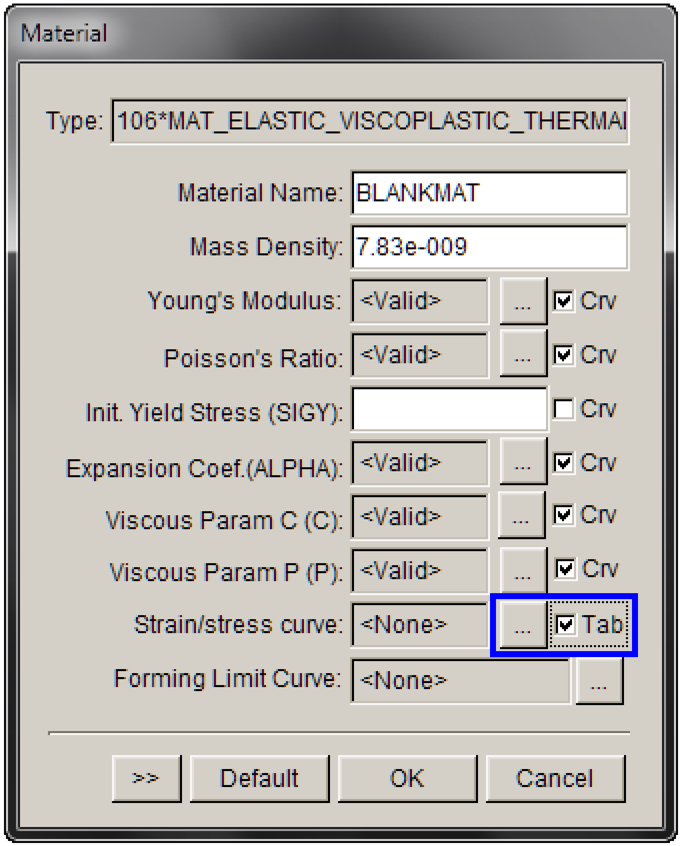

Figure 19

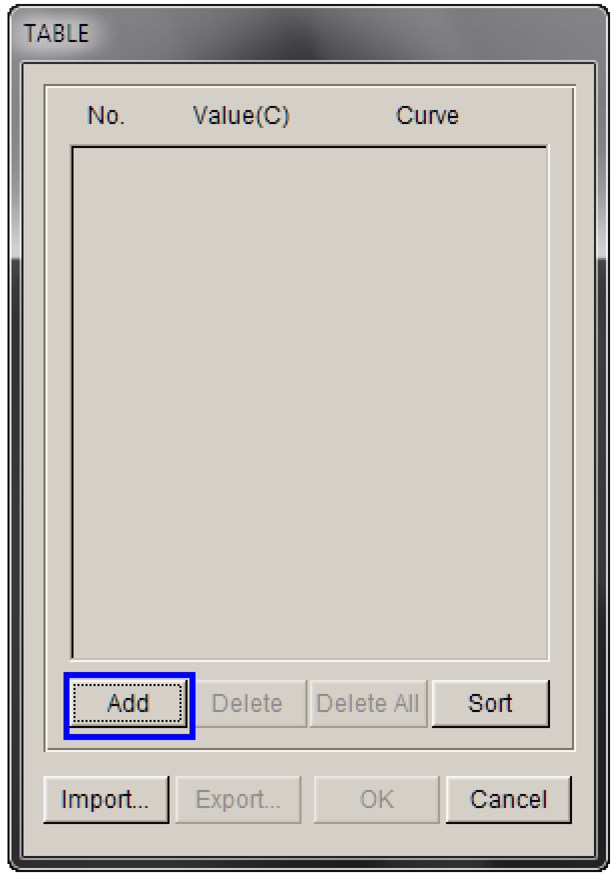

Figure 20

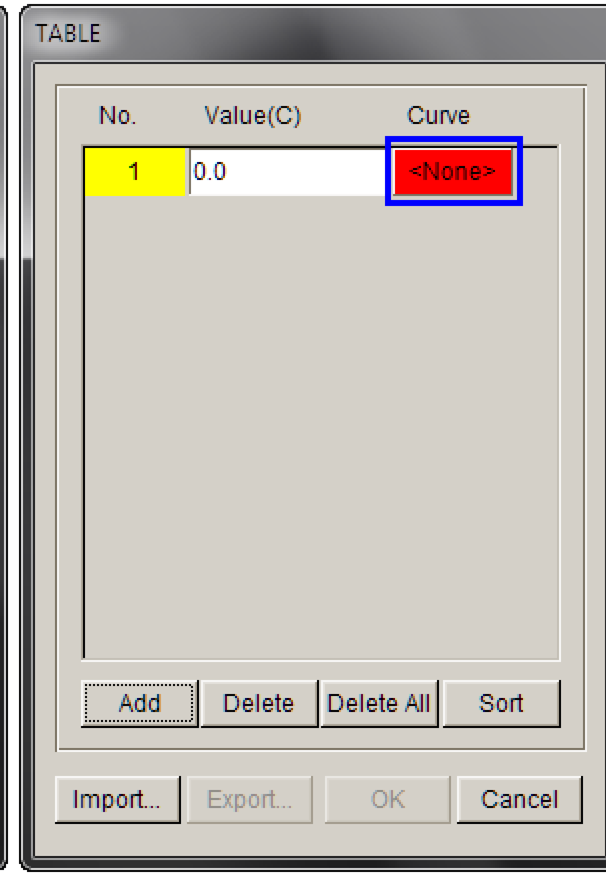

Figure 21

4. Click ***Import…*** to select the stress/strain curve at 0 degree. (Figure 22).
5. Select 0D.cur.
6. Click ***OK***.

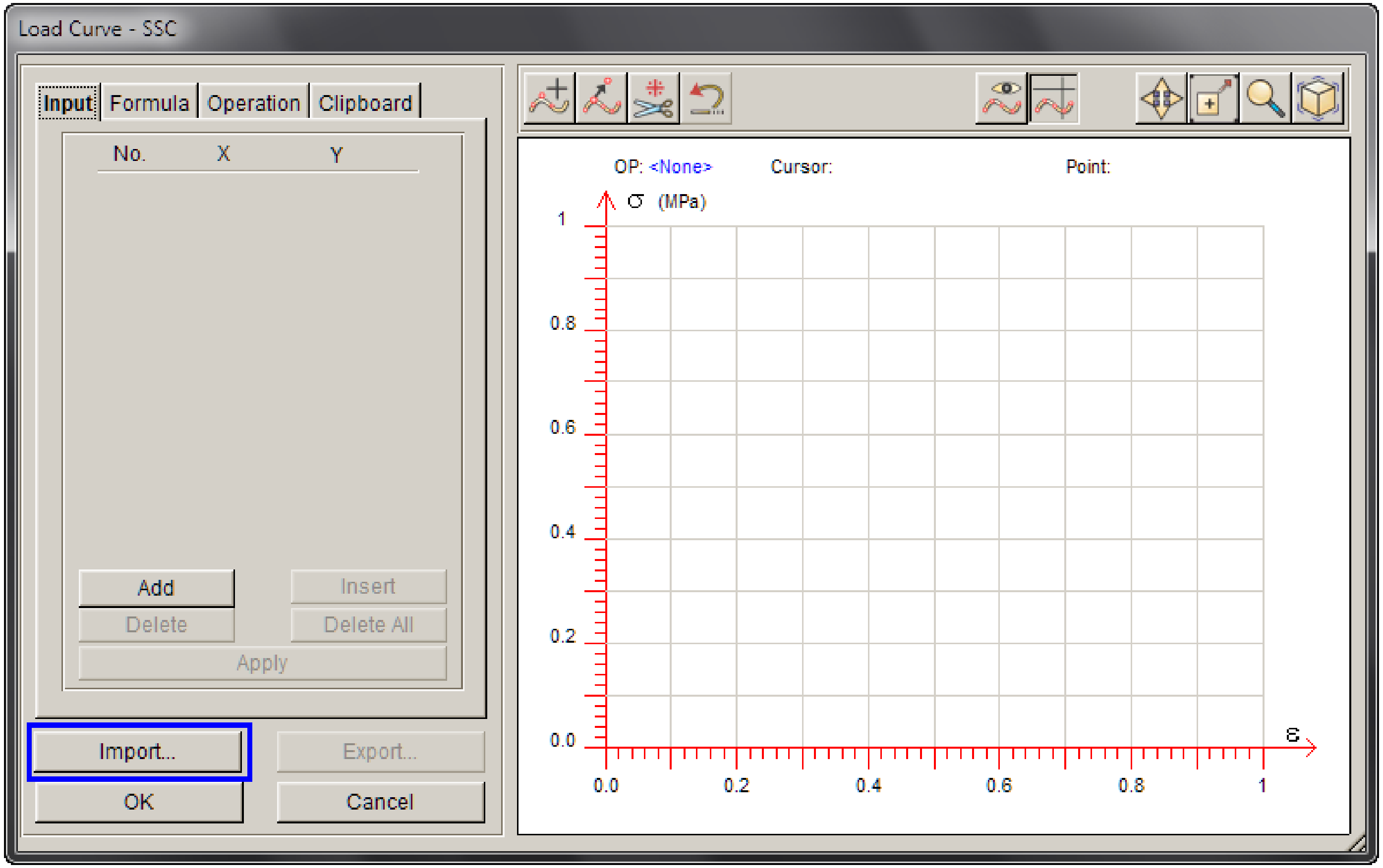

Figure 22

IV. Define blank structural material

7. The stress/strain curve at 0 degree is imported. The result is shown in Figure 23.
8. Then Click ***OK*** to accept the result. The first curve definition is displayed with valid. See Figure 24.

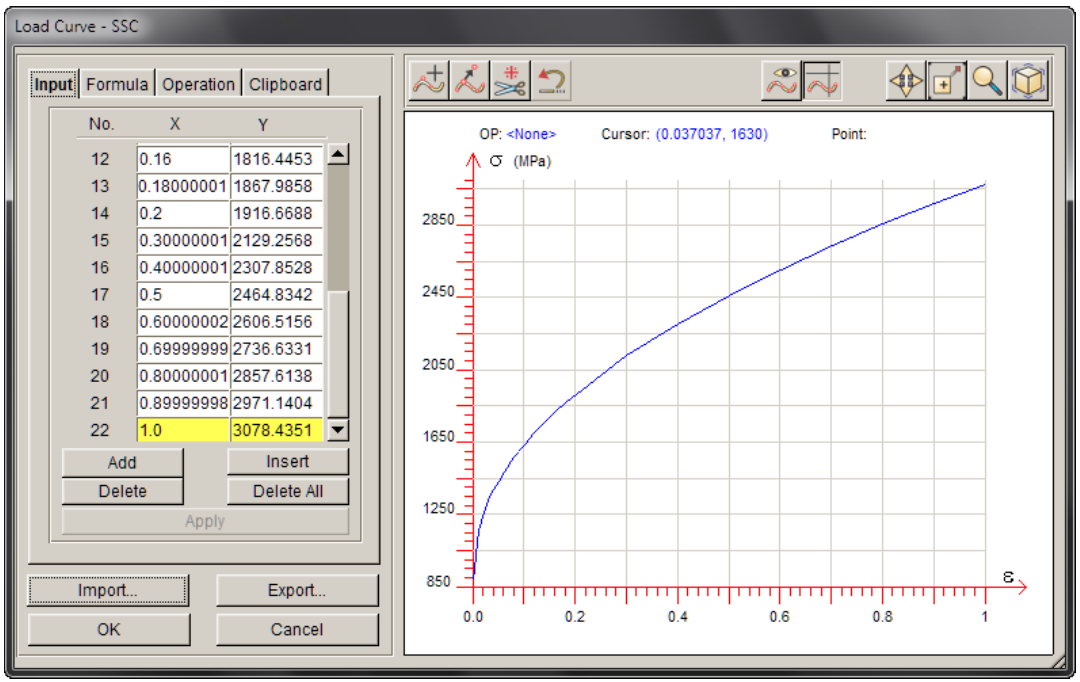

Figure 23

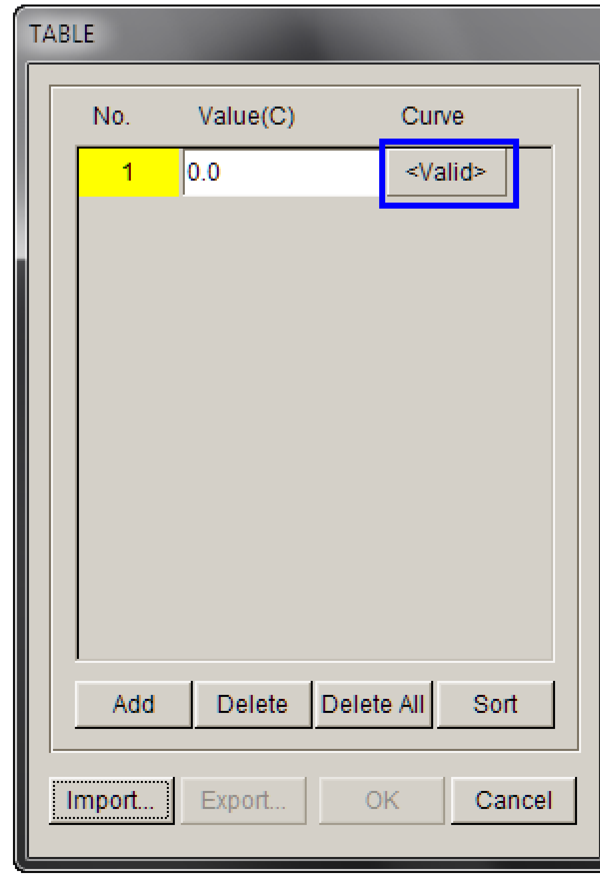

Figure 24

IV. Define blank structural material

9. Repeat the above steps (step 1-step 8) to complete the table.
   - 100D.cur
   - 200D.cur
   - 300D.cur
   - 400D.cur
   - 500D.cur
   - 600D.cur
   - 700D.cur
   - 800D.cur
   - 900D.cur
   - 1000D.cur

   The result is shown in Figure 25.
10. Click ***OK*** to accept the table.
11. Click ***OK*** to accept the stress/strain definition. (Figure 26).

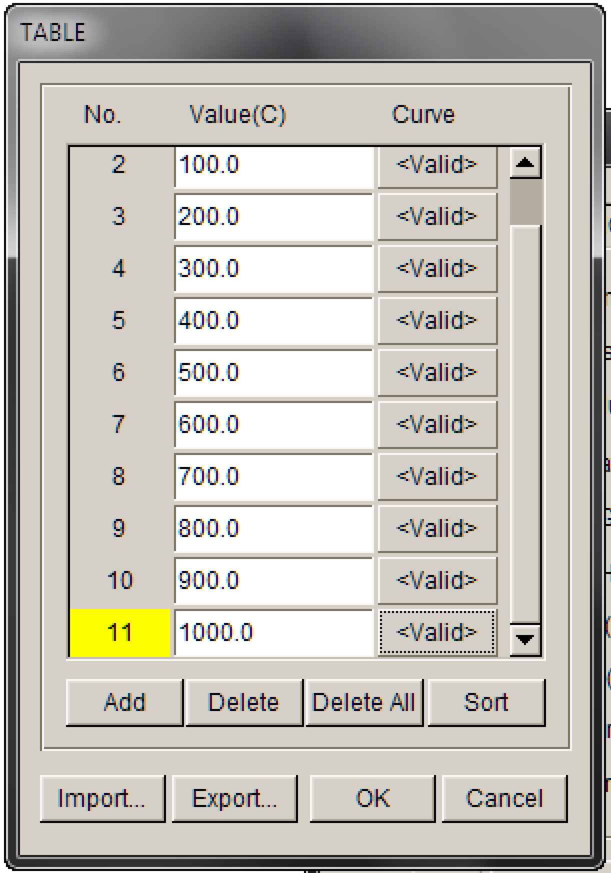

Figure 25

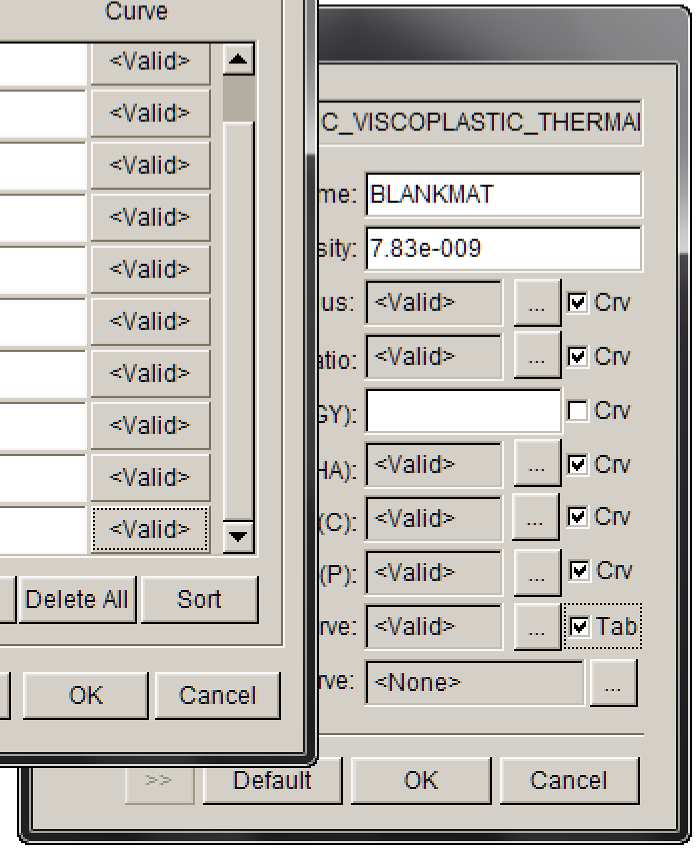

Figure 26

V. Define the symmetry plane

1. Only toggle on the blank part. And switch it to the top view.
2. Click ***Define…*** (Figure 27).
3. The dialog box pops up (Figure 28). Select the Quarter Symmetry.
4. Select the labeled node (Figure 29) in the display screen. Then all nodes on the symmetry plane are selected automatically.
5. Click ***Exit*** in Figure 28 to accept the result.

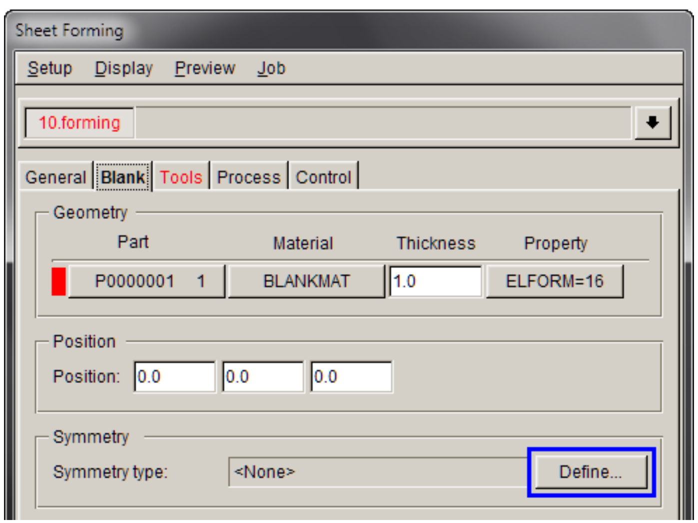

Figure 27

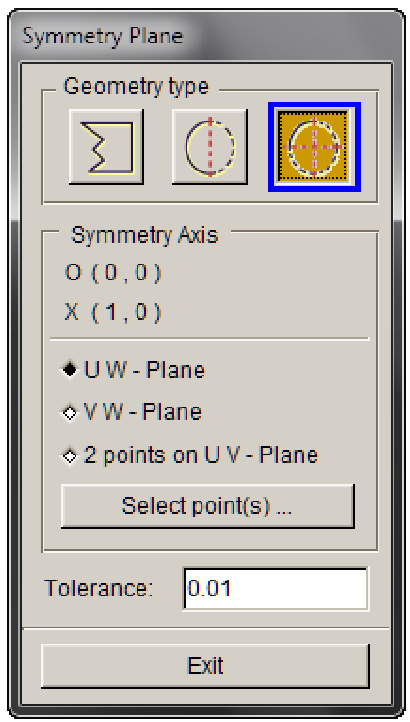

Figure 28

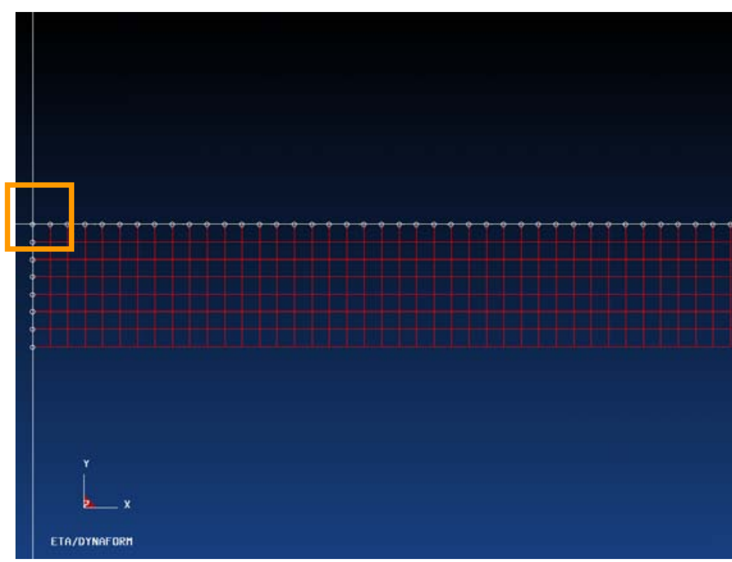

Figure 29

1. Click Tools to define tools. (Figure 30).
2. Click ***Define Geometry…***
3. Click *Define Tool* icon in Tool Preparation dialog box. (Figure 31).
4. Click ***Add Part...*** . See Figure 32.
5. Select part p3 and click ***OK***. (Figure 33).
6. Click ***Exit*** to accept the selected part. (Figure 34).
7. Click ***Exit*** to exit the Tool Preparation dialog box.
8. Repeat steps 1-7 to define punch (p2) and binder (p4).
9. Click ***Exit*** to exit the Tool Preparation dialog box.

(Note: define *Friction Coefficient* as 0.4).

VI. Define tools

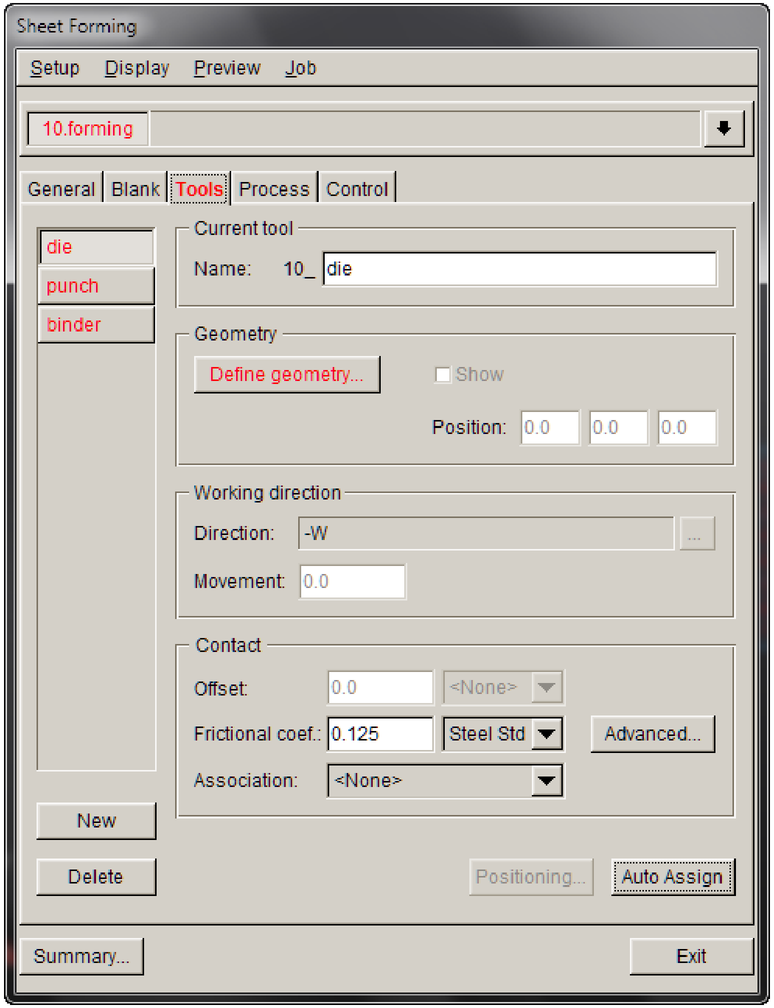

Figure 30

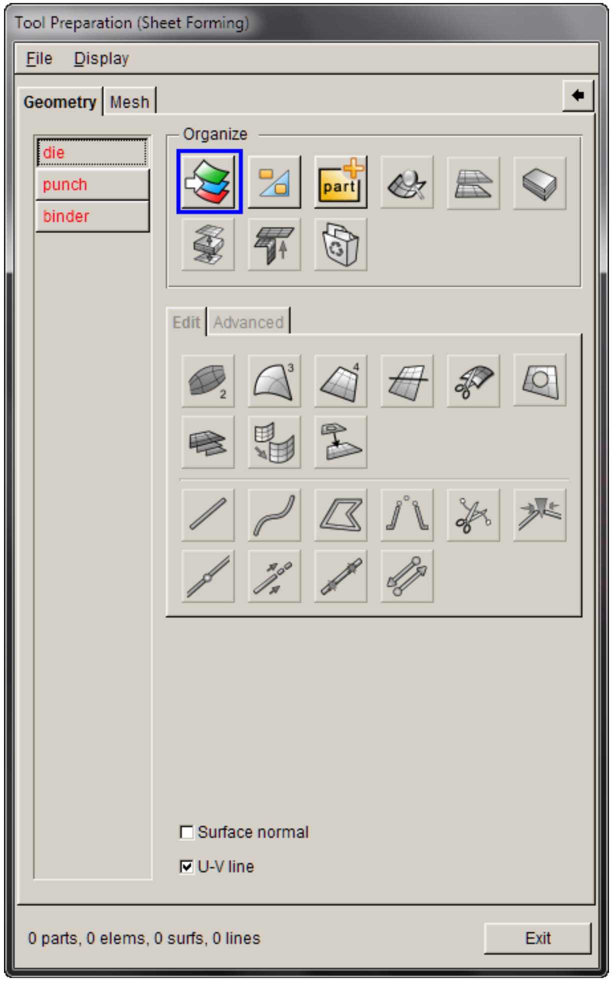

Figure 31

VI. Define tools

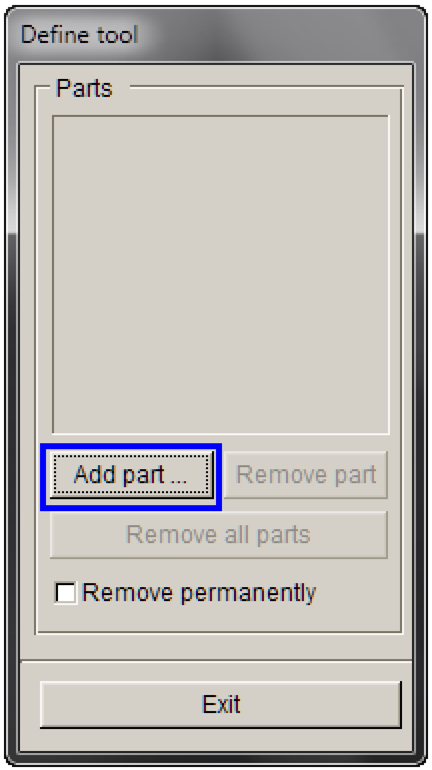

Figure 32

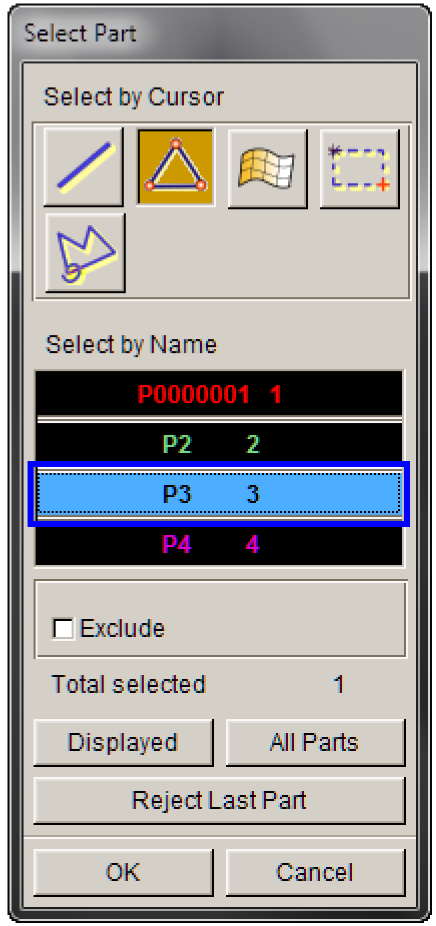

Figure 33

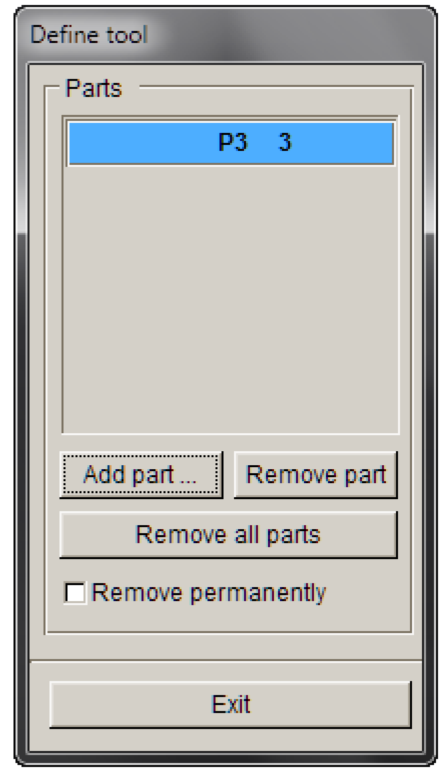

Figure 34

VII. Define thermal analysis

1. Click General on the Sheet forming GUI.
2. Toggle on Coupled thermal structural analysis option.
3. A tab named Thermal is displayed (Figure 35).

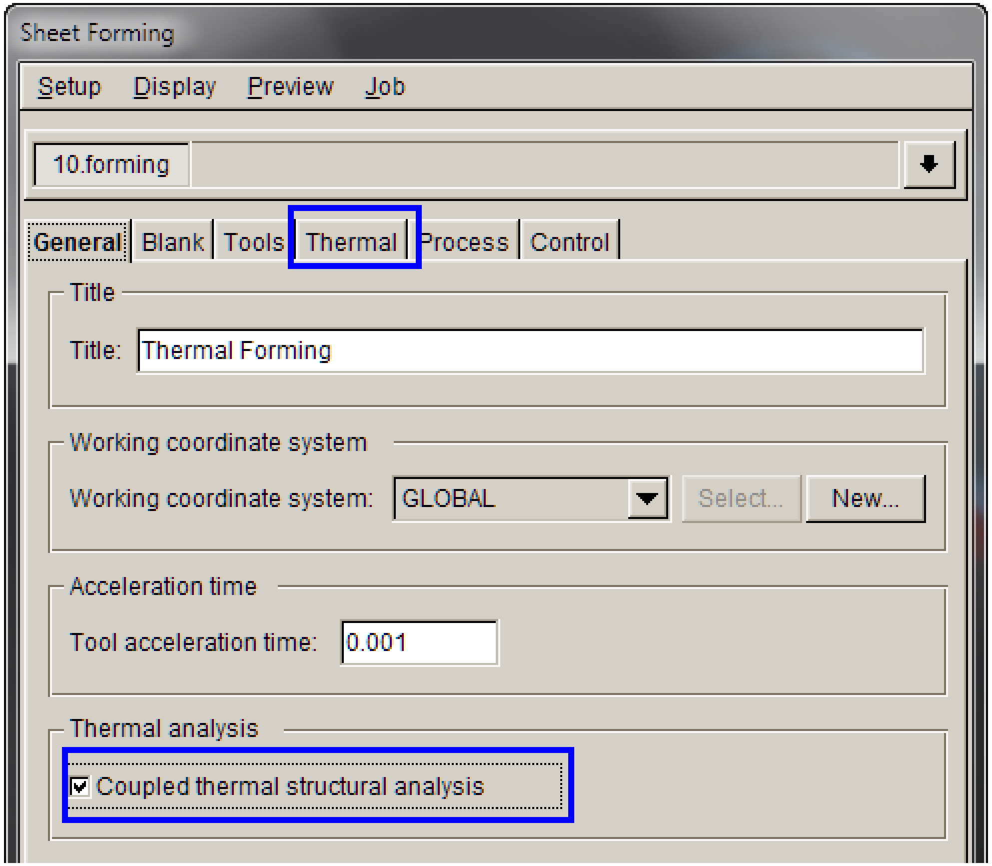

Figure 35

Thermal material

Thermal material type:

*MAT_THERMAL_ISOTROPIC

*MAT_THERMAL_ORTHOTROPIC

MAT_THERMAL_ISOTROPIC_TD

*MAT_THERMAL_ORTHOTROPIC_TD

*MAT_THERMAL_ISOTROPIC_PHASE_CHANGE

MAT_THERMAL_ISOTROPIC_TD_LC

MAT_ADD_THERMAL_EXPANSION

VIII. Define thermal material

1. Click Blank on the Thermal analysis tab.
2. Define the blank thermal material and select *MAT_TERMAL_ISOTROPIC_TD_LC. (Figure 36).
3. Use Thermal Density default (same as structure density).
4. Click ***…*** to define Heat Capacity. (Figure 37).
5. Click ***Import…*** to import the heat capacity curve. (Figure 38).

VIII. Define thermal material

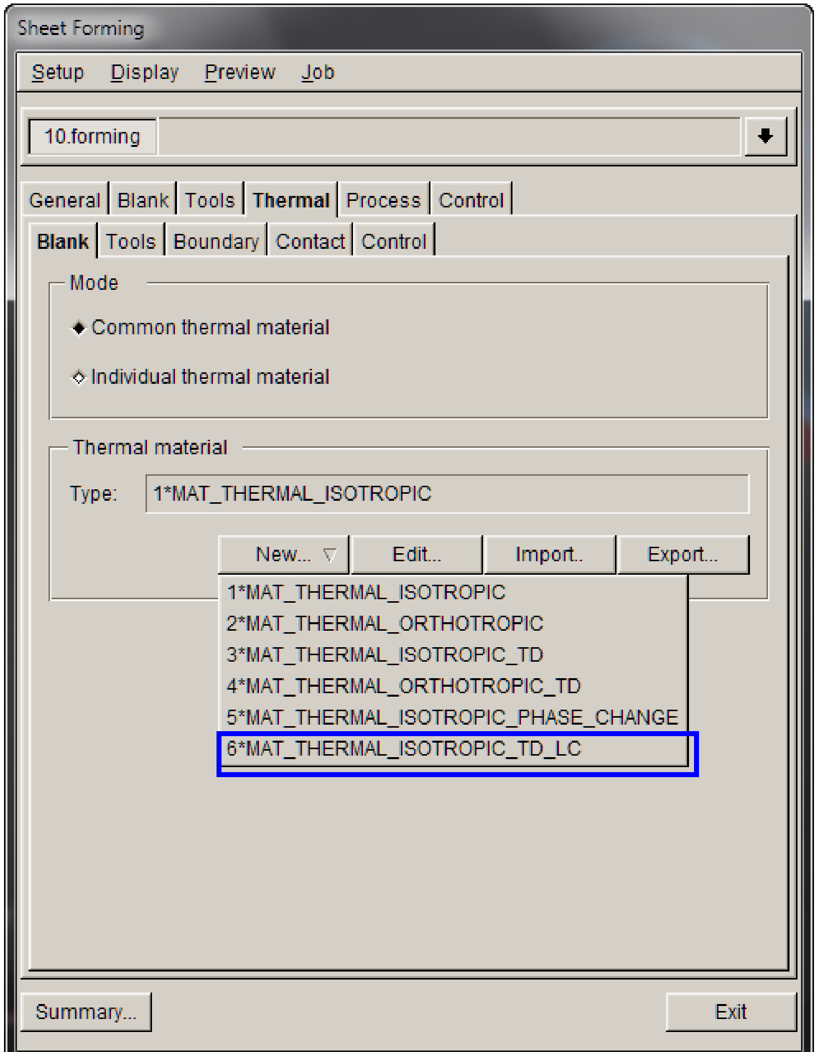

Figure 36

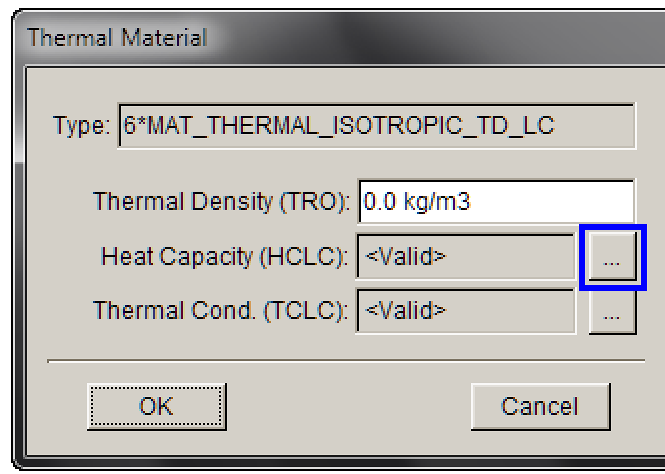

Figure 37

VIII. Define thermal material

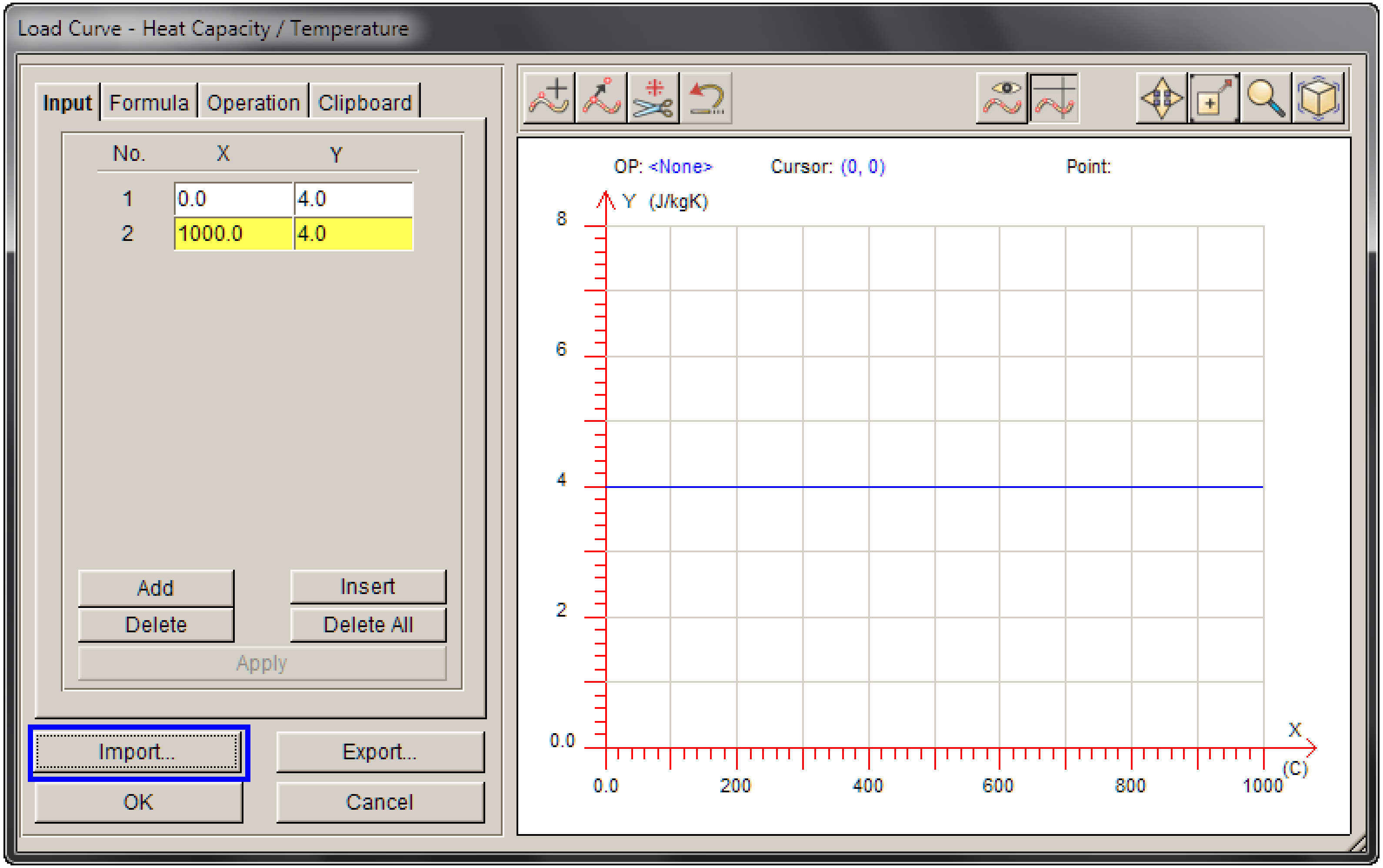

Figure 38

VIII. Define thermal material

6. Select HCLC-heat-capacity.cur.
7. Click ***OK*** to accept the curve.
8. The file is imported. (Figure 39). (The unit of heat capacity here is J/kgK.)
9. Click ***OK*** to accept the operation.

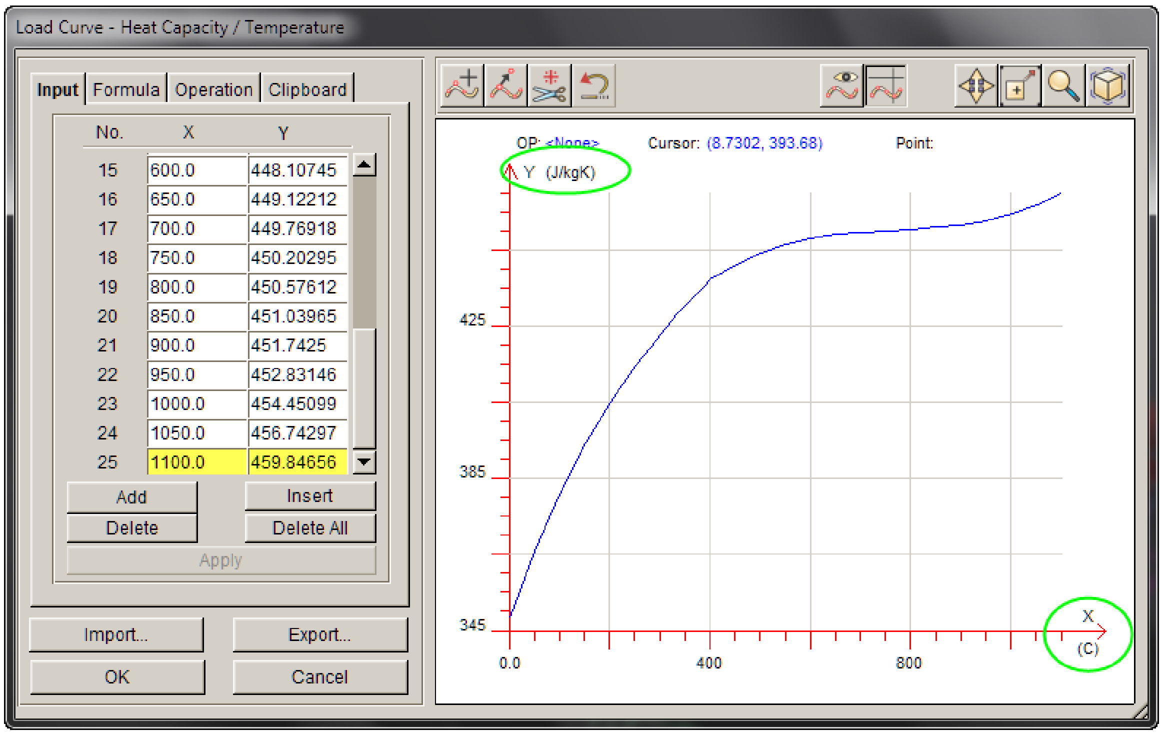

Figure 39

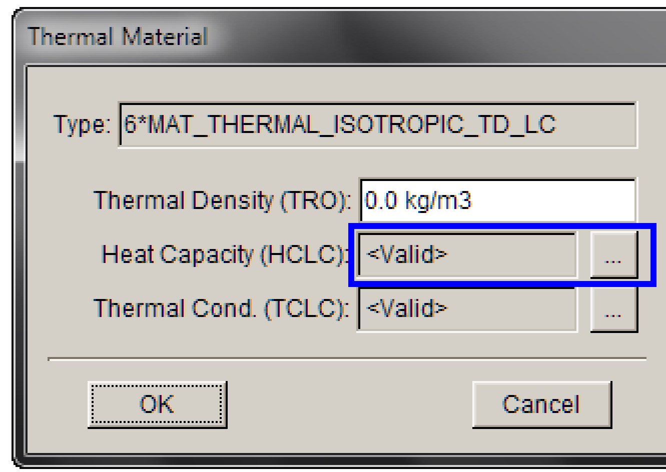

Figure 40

VIII. Define thermal material

10. Repeat steps 4-9 to select TCLC-conductivity-scaled.cur as blank thermal conductivity curve. (Figure 41).
11. The result is shown in Figure 42.

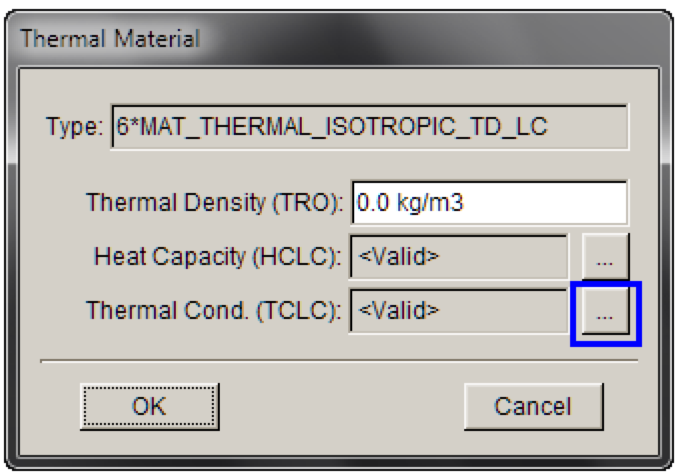

Figure 41

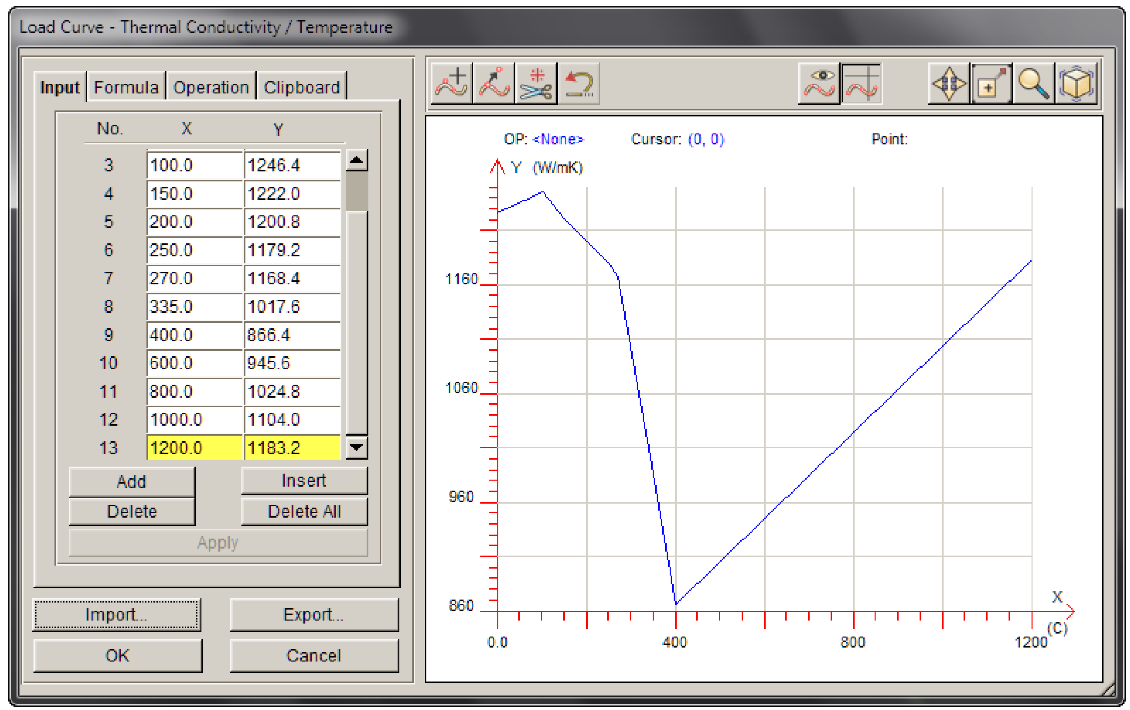

Figure 42

VIII. Define thermal material

12. Refer to the above steps to define the tools thermal material.
    - HCLC-heat-capacity-tools.cur
    - TCLC-conductivity-tool-scaled.cur

    The result is shown in Figure 43.

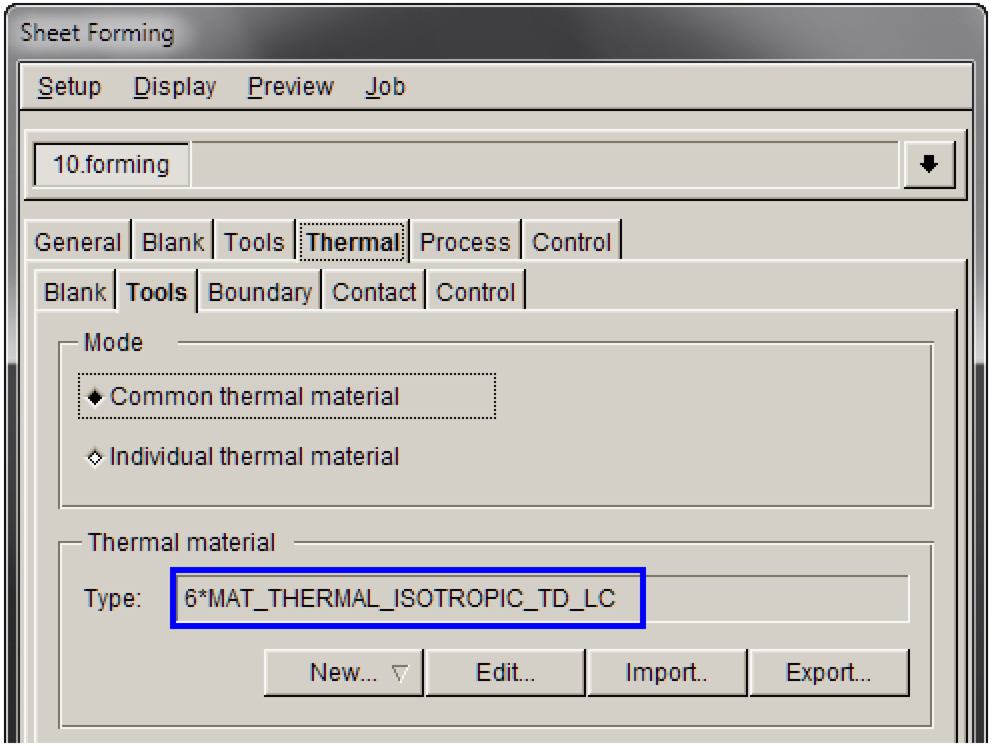

Figure 43

IX. Define thermal boundary

1. Keep the initial blank temperature and tools temperature as default (800 and 50).
2. Keep the Radiation factor as default (0.8*SBC and SBC=5.67E-11).
3. Keep the Heat transfer coefficient as default (5W/m2K).

   Please see Figure 44.

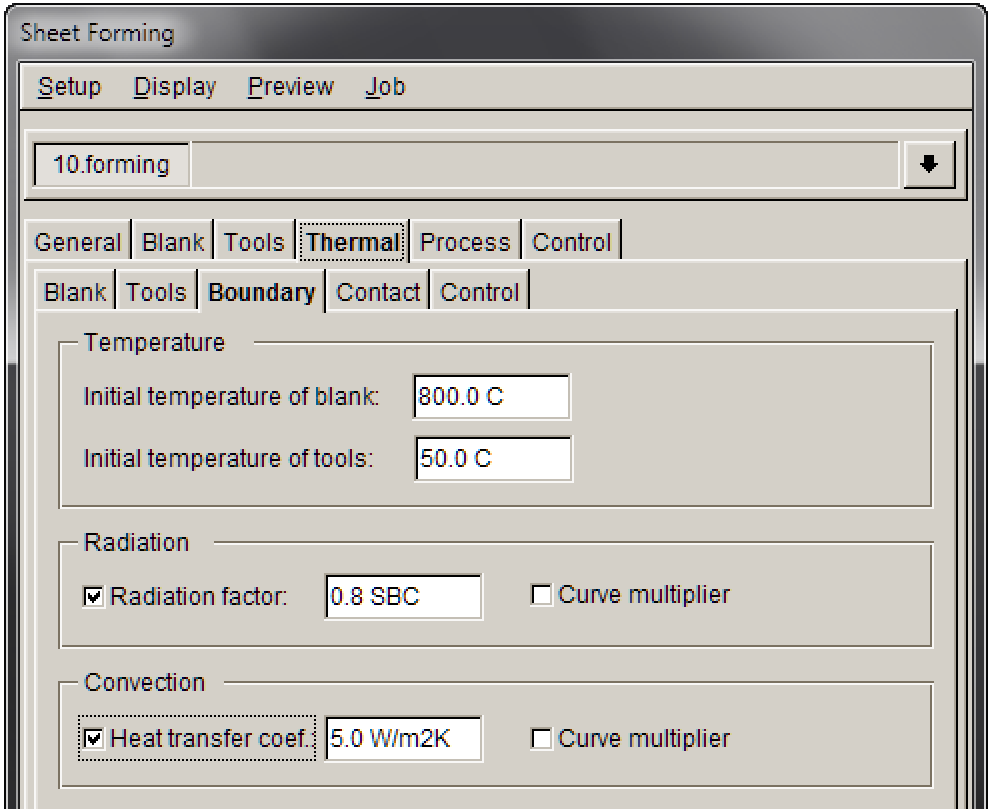

Figure 44

X. Define thermal contact

1. Define Thermal Conductivity as 40W/mK;
2. Define Radiation factor as 7.6*SBC;
3. Define Heat Transfer Coefficient as 20W/m2K;
4. Min. gap is 0.001mm and Max.gap is 0.5mm.
5. Select Two Way contact.
6. Toggle on Thermal boundary conditions are off when parts are in contact.
7. The definition is shown in Figure 45.

X. Define thermal contact

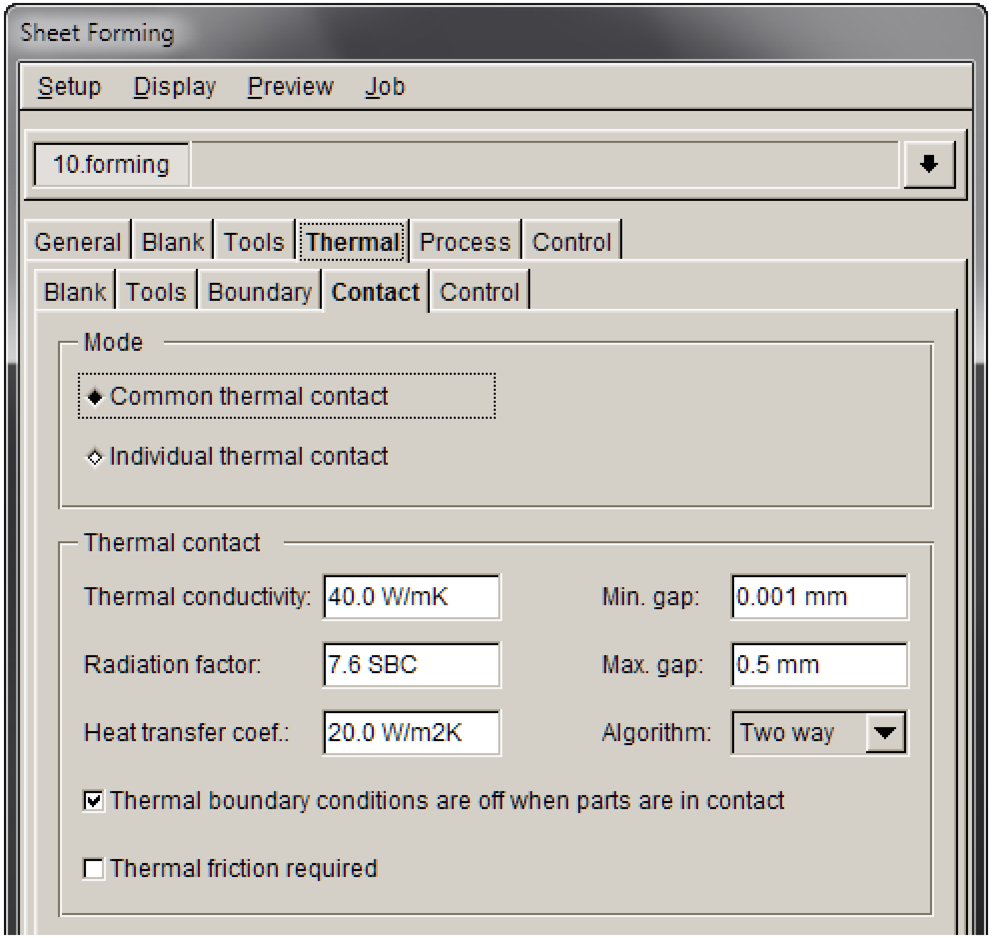

Figure 45

XI. Define thermal control

1. Define Thermal analysis type: 1 (0, Steady state analysis; 1, transient analysis.) Please see Figure 46.
2. Define Thermal problem type: 1 (Nonlinear problem with material properties evaluated gauss point temperature).
3. Click ***Advanced*** to define all parameters illustrated in Figure 47.

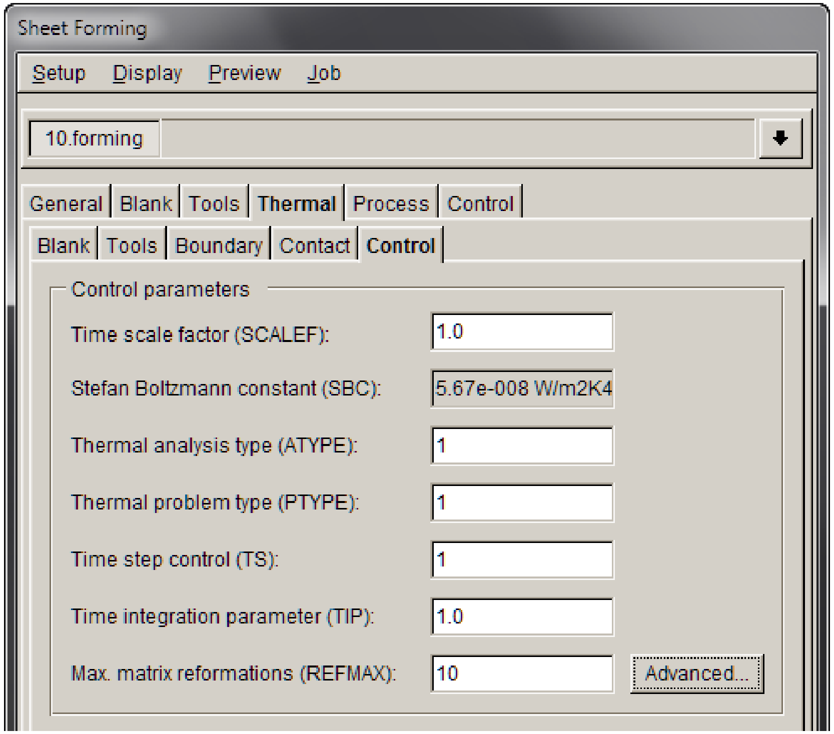

Figure 46

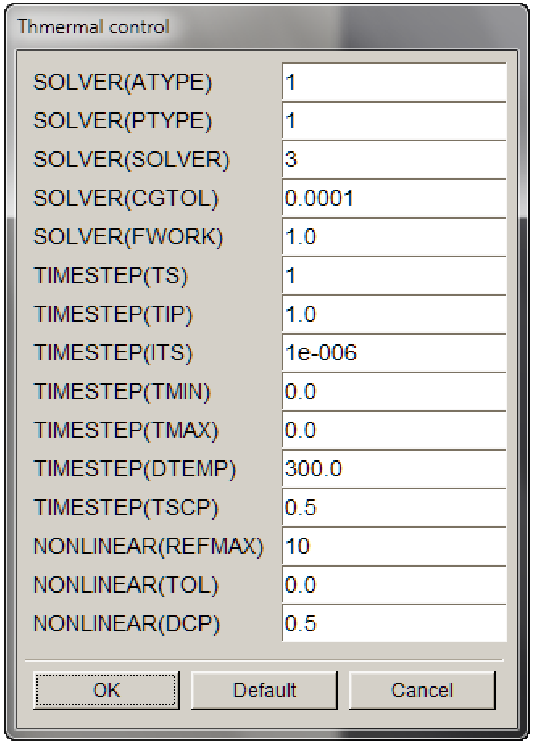

Figure 47

XII. Define process

1. Click Process in Figure 48.
2. Select velocity curve as Sinu.W-H. And Keep Closing step default (Figure 48).
3. Drawing step is defined as illustrated in Figure 49.

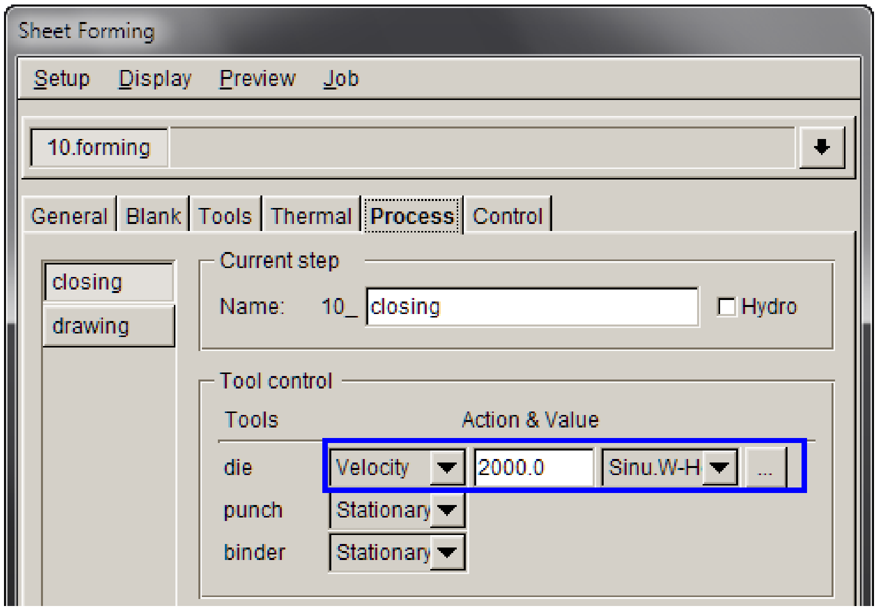

Figure 48

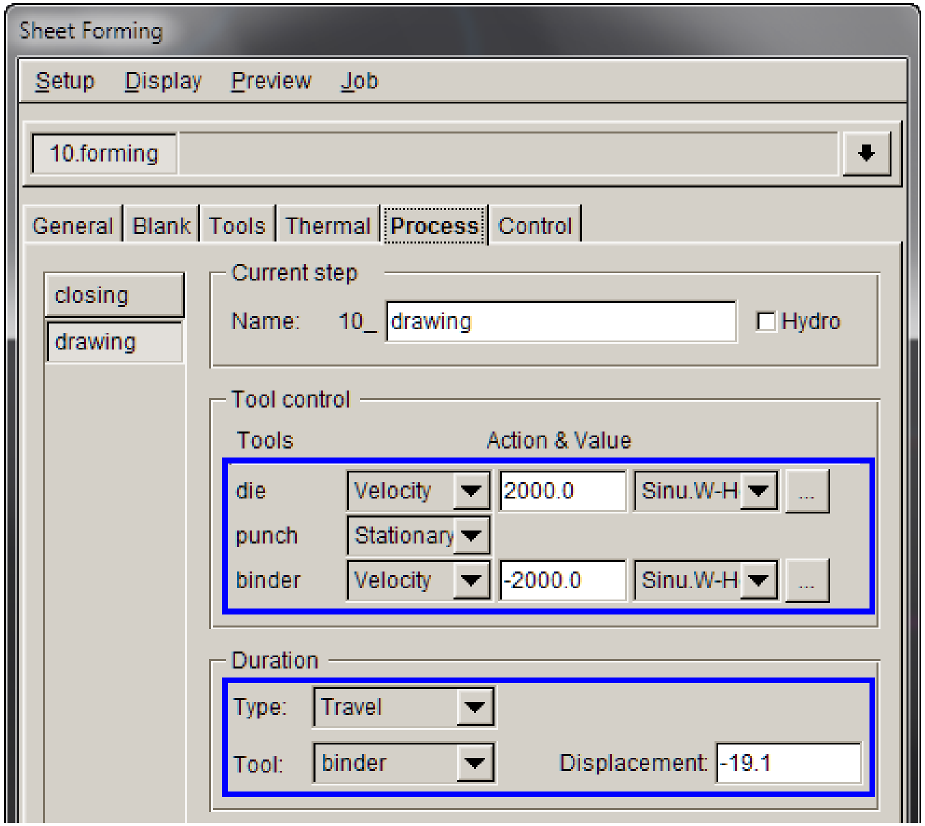

Figure 49

XIII. Define control

1. Click Control to define control parameters. See Figure 50.

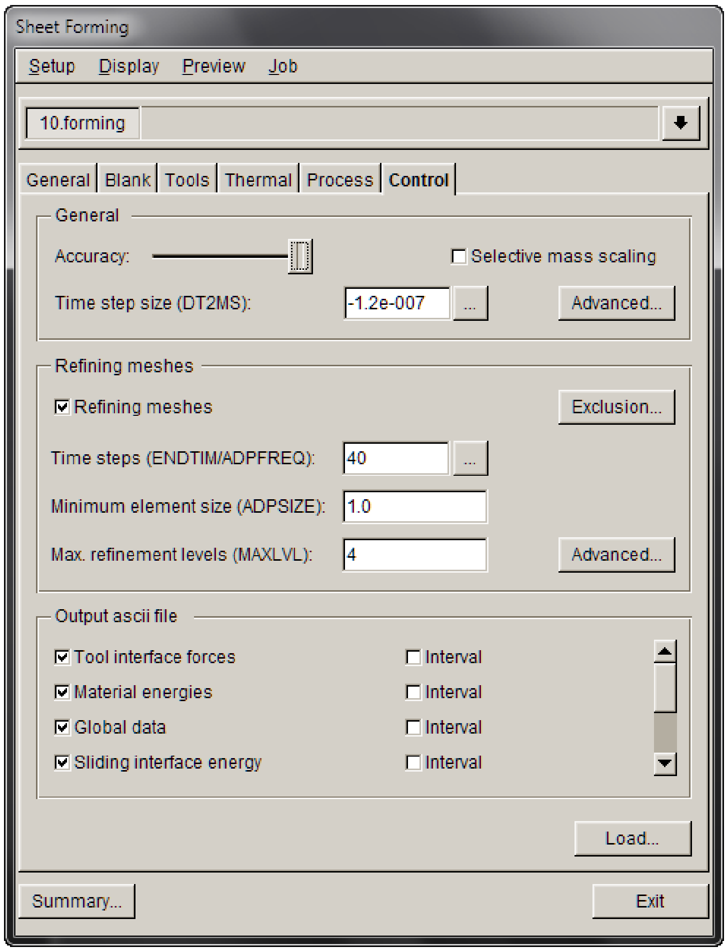

Figure 50

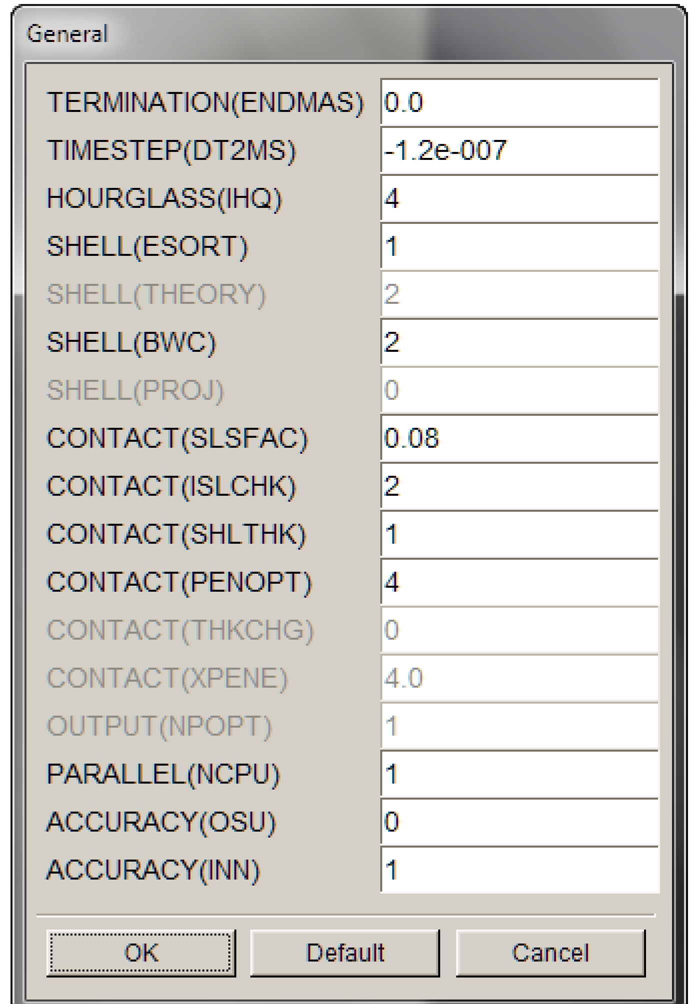

Figure 51

XIV. Add a cooling stage

1. Click New to add a new stage. See Figure 52.
2. Select the type: Cooling. See Figure 53.

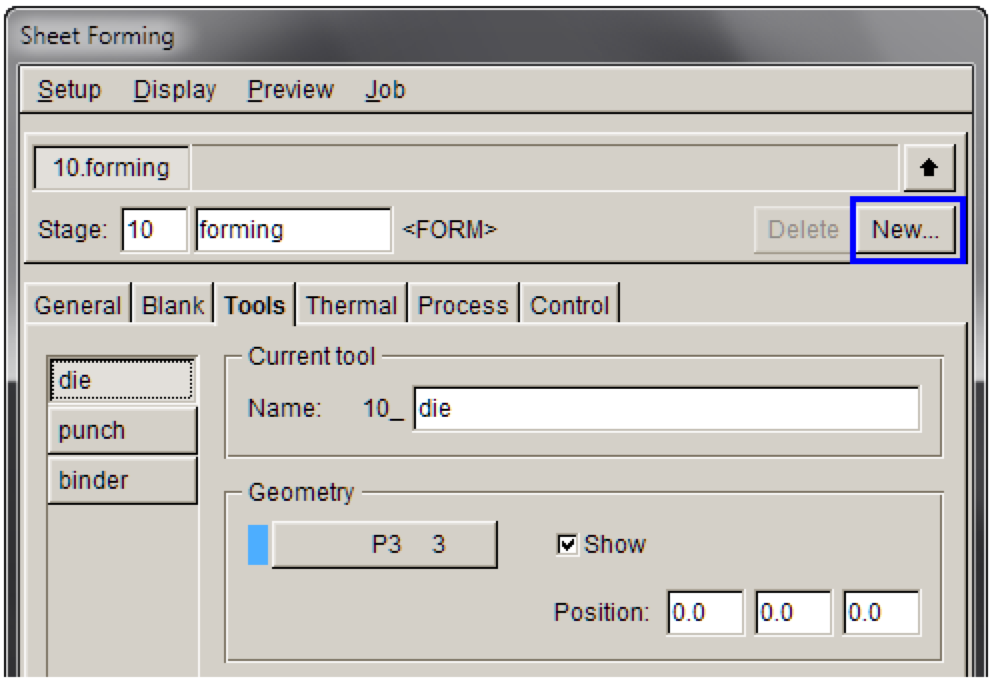

Figure 52

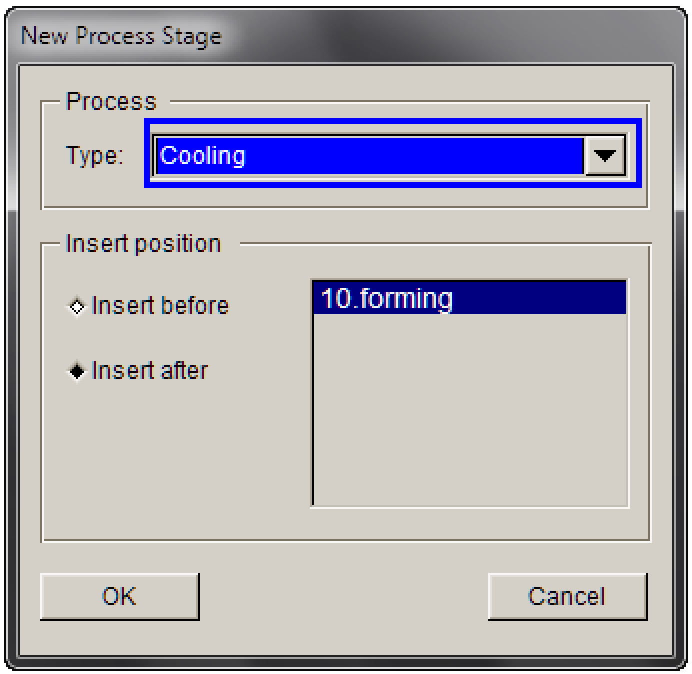

Figure 53

XV. Define tools

1. Click New to add tool: punch.
2. Click *Apply*.
3. Click New to add tool: binder.
4. Click *Apply*.

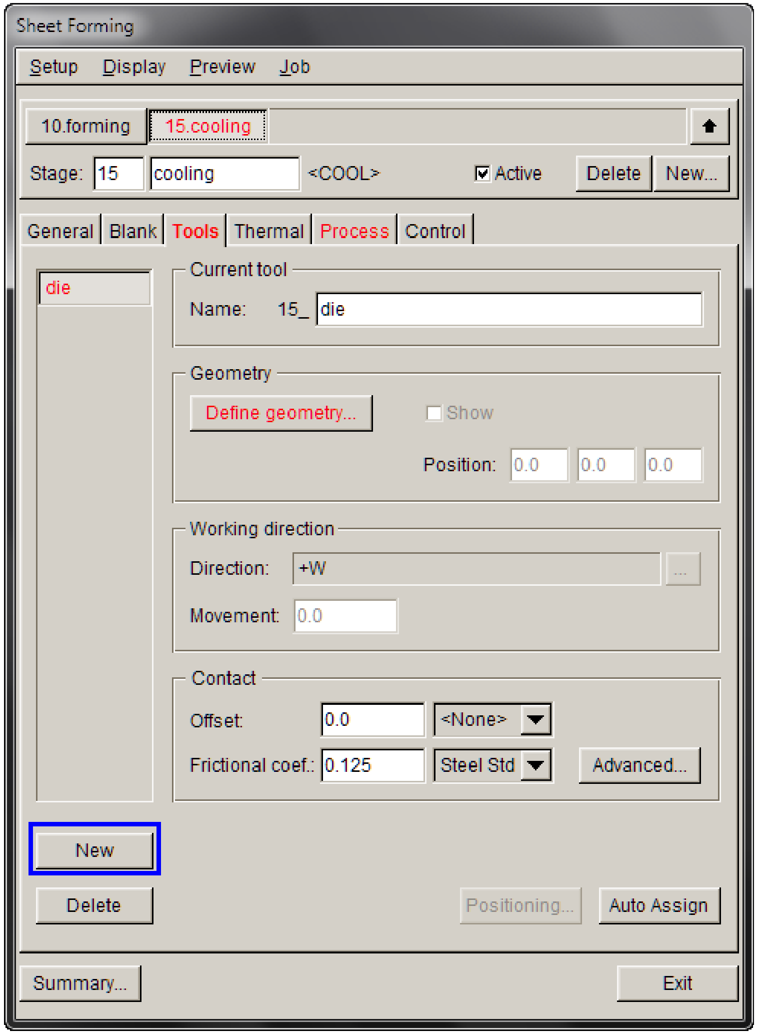

Figure 54

XV. Define tools

1. Click ***Define Geometry…***
2. Select the tool from the list: 15_die.
3. Click *Define Tool* icon in Tool Preparation dialog box. (Figure 55).
4. Click ***Add Part...*** . (Figure 56).
5. Select part p3 and click ***OK***. (Figure 57).
6. Click ***Yes*** to copy the selected part. (Figure 58).
7. Click ***Exit***.
8. Repeat steps 2-7 to define 15_punch (P2) and 15_binder (P4).
9. Click ***Exit*** to exit the Tool Preparation dialog box.

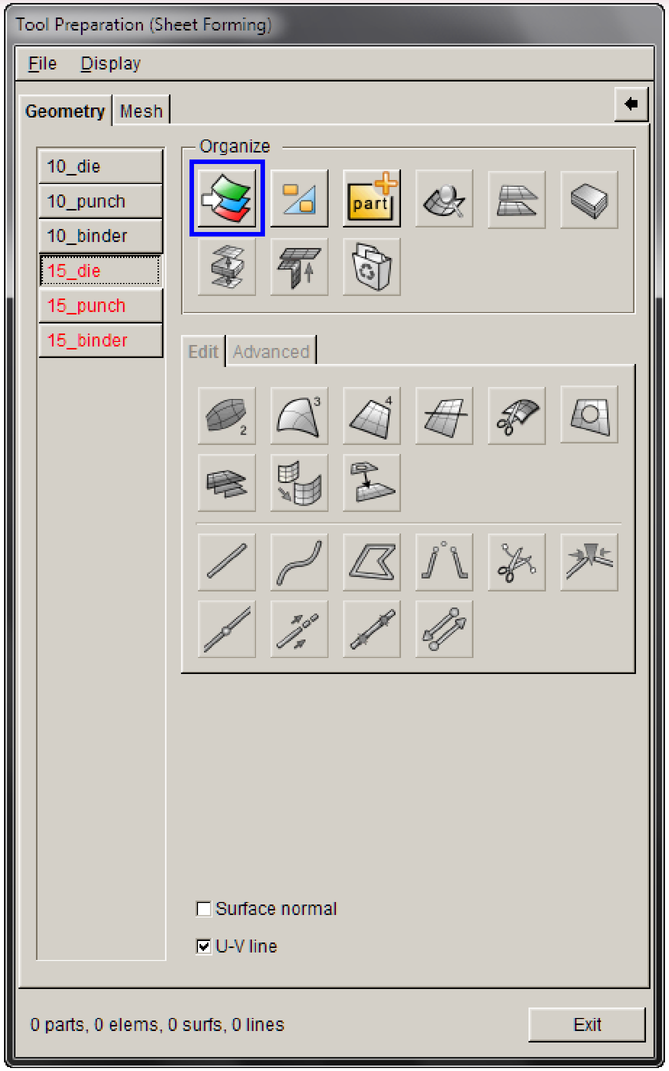

Figure 55

Figure 56

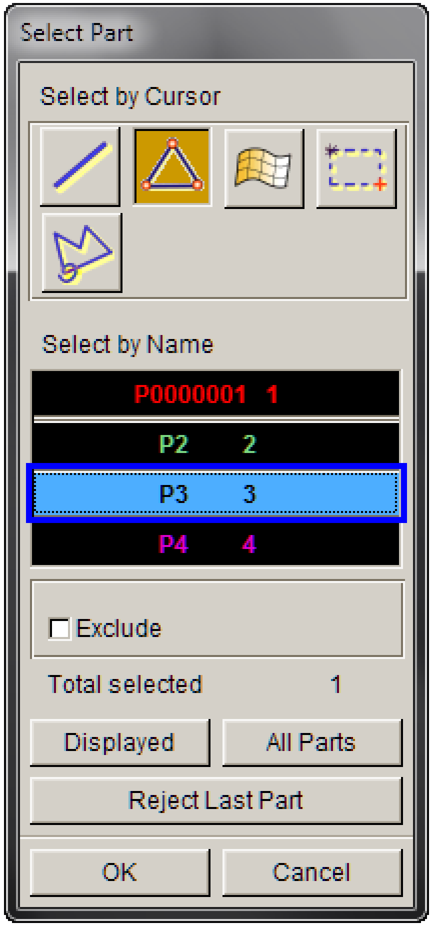

Figure 57

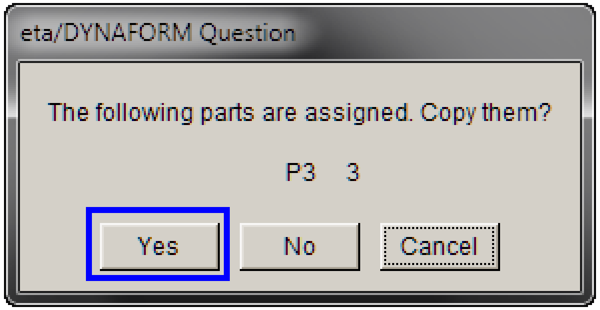

Figure 58

XV. Define tools

1. Click ***Positioning…*** to define the tools position.
2. Key in the movement of the tools. (Figure 59).
3. Click OK to exit.

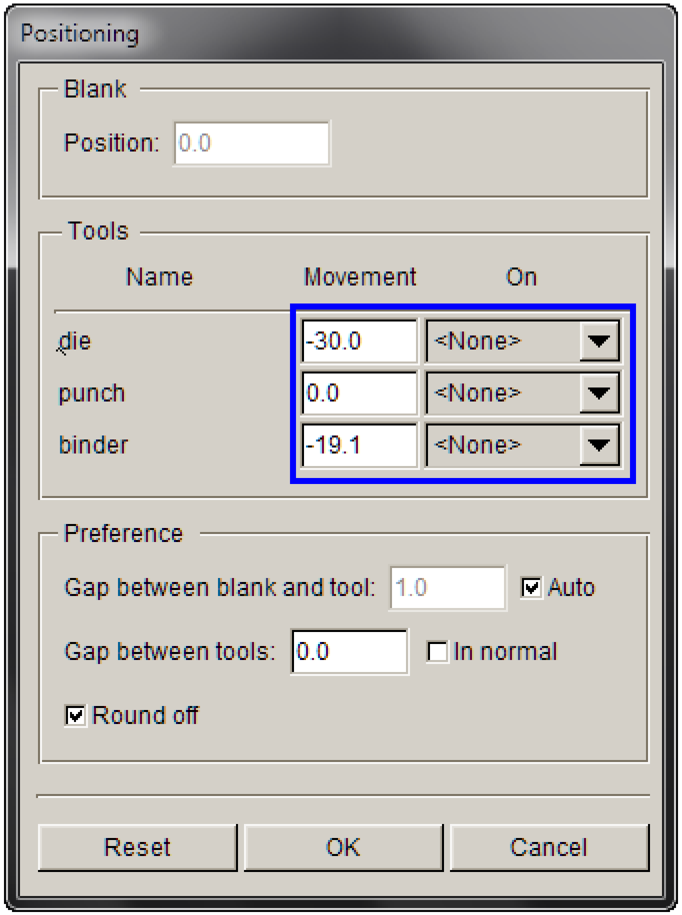

Figure 59

XVI. Define thermal parameters

1. Define the thermal material, contact and control parameters same with the forming stage.

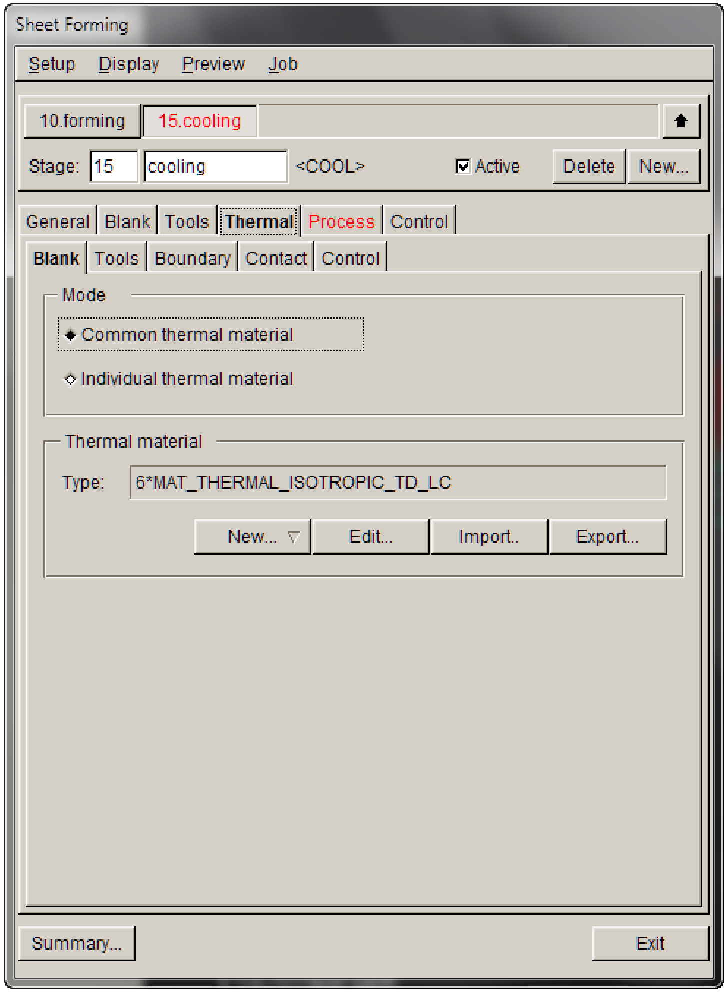

Figure 60

XVI. Define thermal parameters

2. Toggle off Radiation and Convection option in cooling stage.

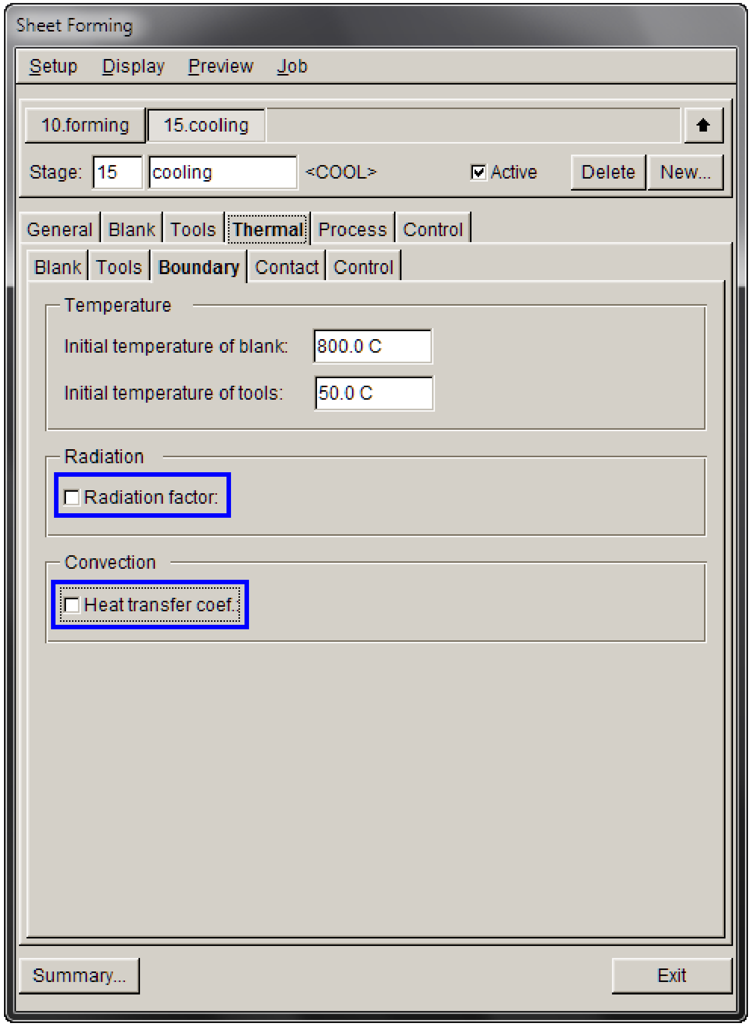

Figure 61

XVII.Define cooling process

1. Click Process to define the process parameters.
2. Key in the duration time: 1

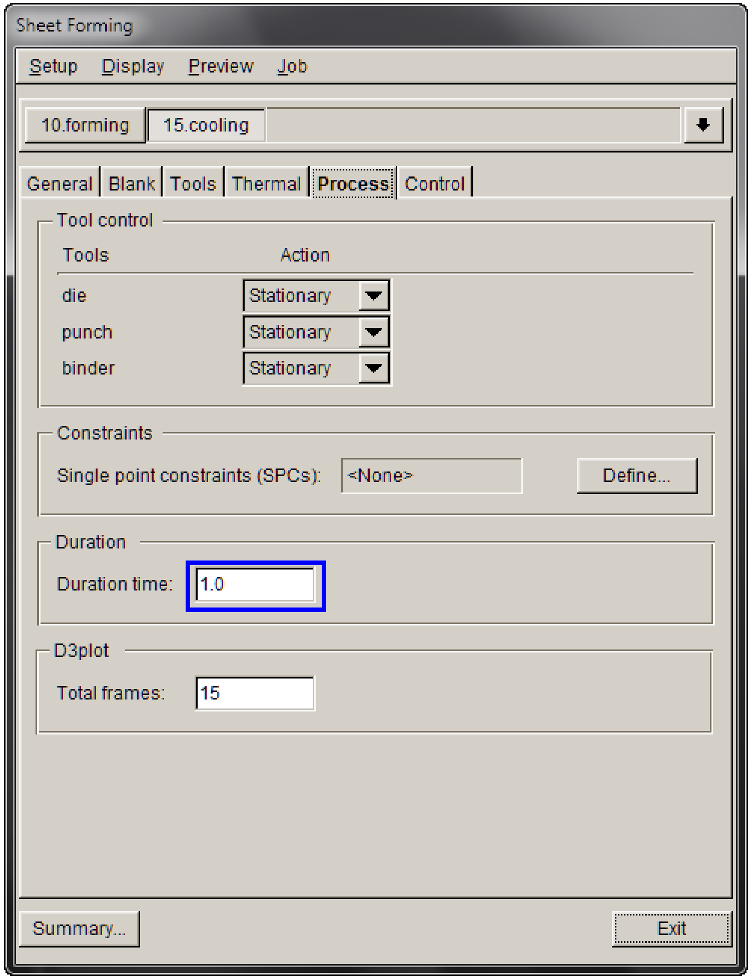

Figure 62

1. Click the menu Job/Job Submitter to submit the job.
2. The LS-DYNA solver starts automatically.

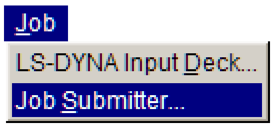

Figure 63

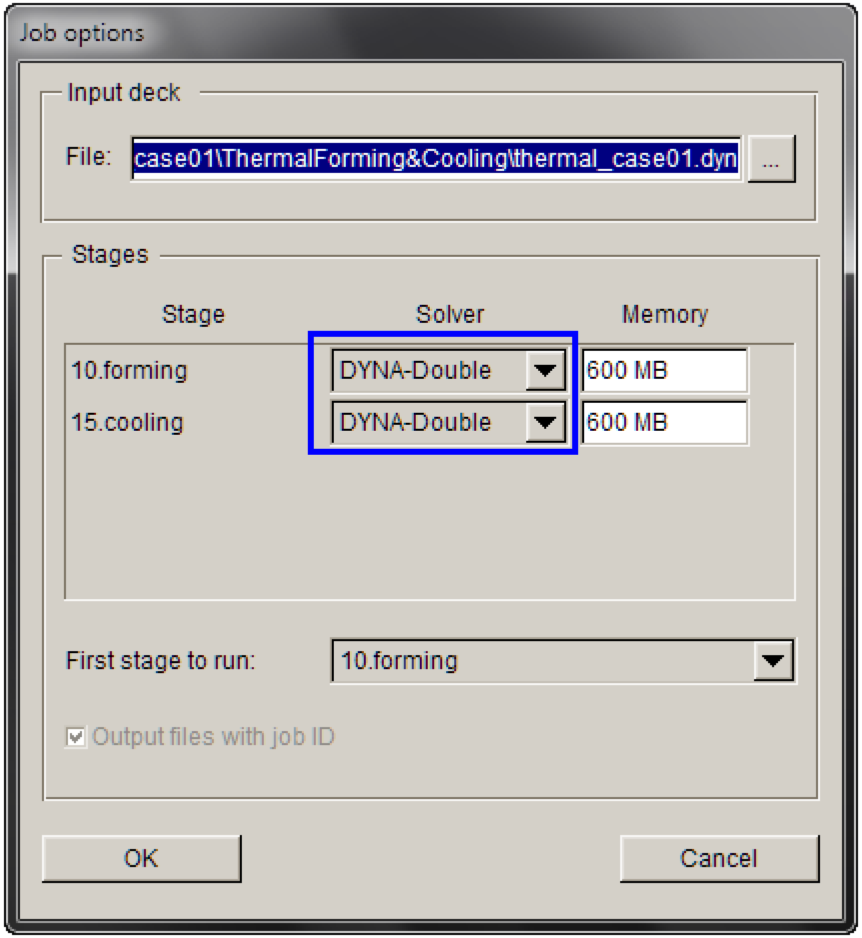

Figure 64

XIX.View the result in eta/POST

1. Click eta/POST from eta/DYNAFORM menu.
2. Open the result of thermal forming.
3. Click the menu: Application->Show Blank Part Only.
4. Click the contour icon  .
5. Select the Temperature as contour plot. (User can switch the temperature unit with Kelvin and Celsius).

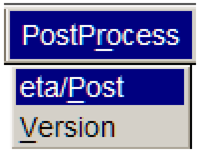

Figure 65

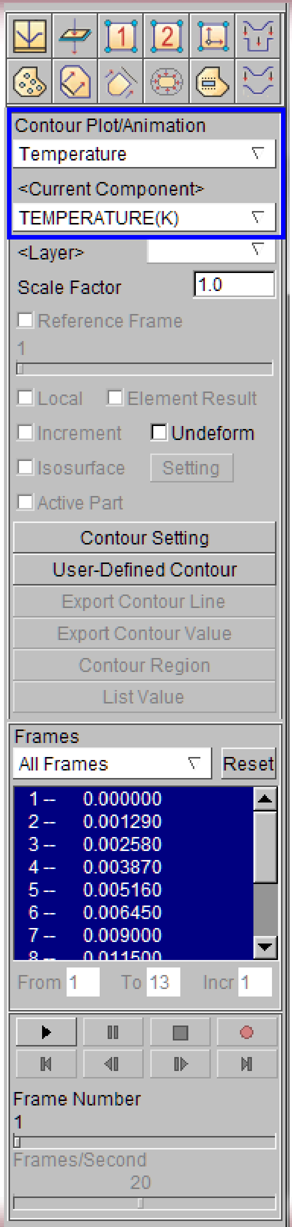

Figure 66

XIX.View the result in eta/POST

Temperature contour plot

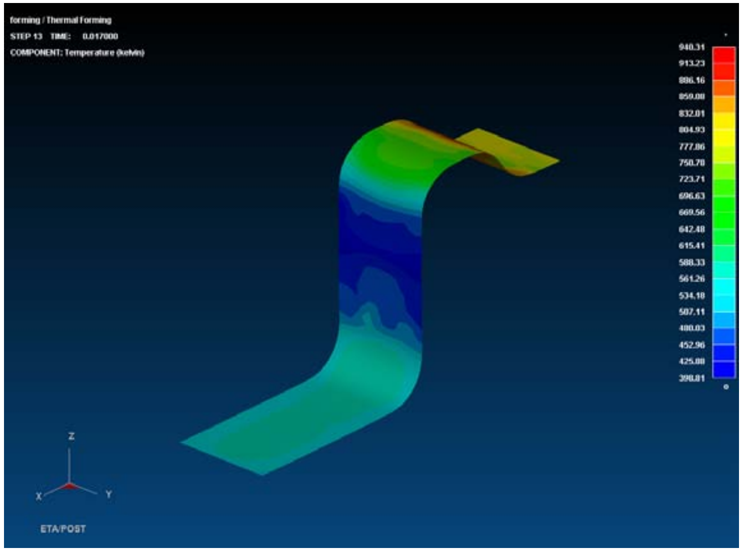

Figure 67

Last frame of forming

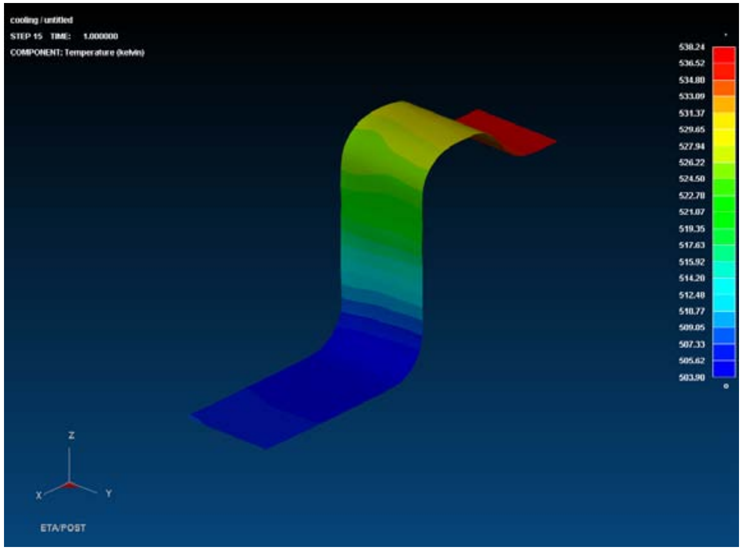

Figure 68

Last frame of cooling