Tutorial

Simulation of a heat source in gas metal arc welding by using LS-PrePost®

written by:

Braj Bhushan Prasad

Technische Universität Berlin
Fakultät Verkehrs- und Maschinensysteme
Institut für Mechanik

Contents

List of Figures

1 Introduction

LS-DYNA® is a finite element code released by Livermore Software Technology Corporation (LSTC) for transient dynamic analysis capable of simulating complex real world problems. For solving coupled multi-physics and multi-stage problems, LS-DYNA® provides solvers, like arbitrary LAGRANGian EULERian (ALE), coupled fluid dynamics (CFD), smooth particle hydrodynamics (SPH), *etc.*

LS-DYNA® include a pre-processing and postprocessing environment for the analysis of finite element models. It is used in a wide range of industries like automotive, aerospace etc., and also is extensively used in academic and research institutions due to its capability to address non-linear problems. The LS-PrePost® graphical user interface (GUI) is shown in Fig. 1.1. Fig. 1.1 helps the user to get familiar with the icons.

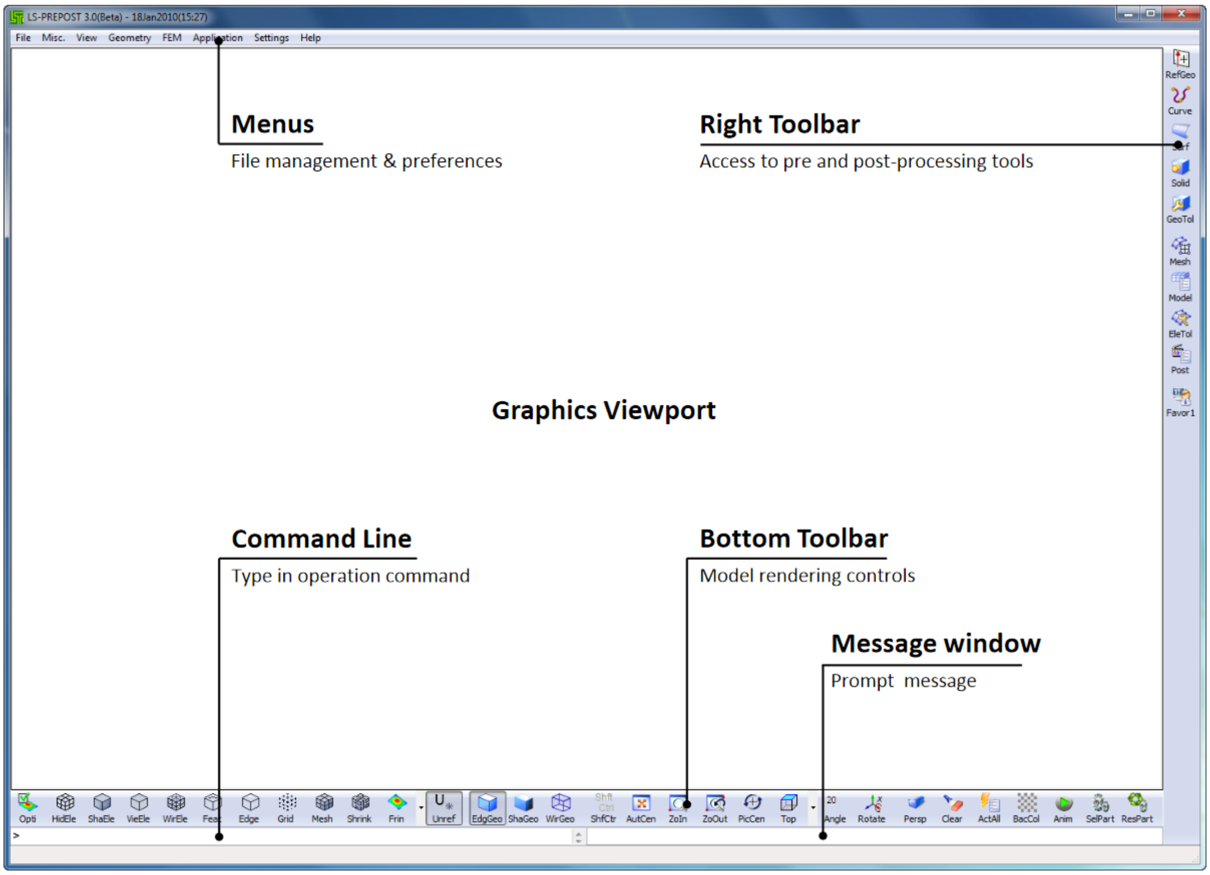

Figure 1.1: Graphical user interface (GUI) of LS-PrePost

The complete finite element analysis (FEA) is done in three different stages, namely preprocessing, analysis, and postprocessing, respectively. During preprocessing the given physical problem is modeled to generate a keyword file (.k). In the second stage, *i.e.* in analysis the modeled physical problem is solved numerically. The duration of the analysis depends on the complexity of the given problems. In the last stage the results are evaluated by using a binary output database file (d3plot), see Fig. 1.2.

The general approach involved in solving physical problems by using LS-DYNA® are mentioned below:

- Create finite element (FE) Model
- Choose Material Model and Properties
- Assign Material and Property
- Assign loads and boundary condition
- Specify control parameters
- Create ".k" input file
- Solve the .k file in LS-DYNA® solver to get "d3plot" output file
- Post process the d3plot file in LS-PrePost®

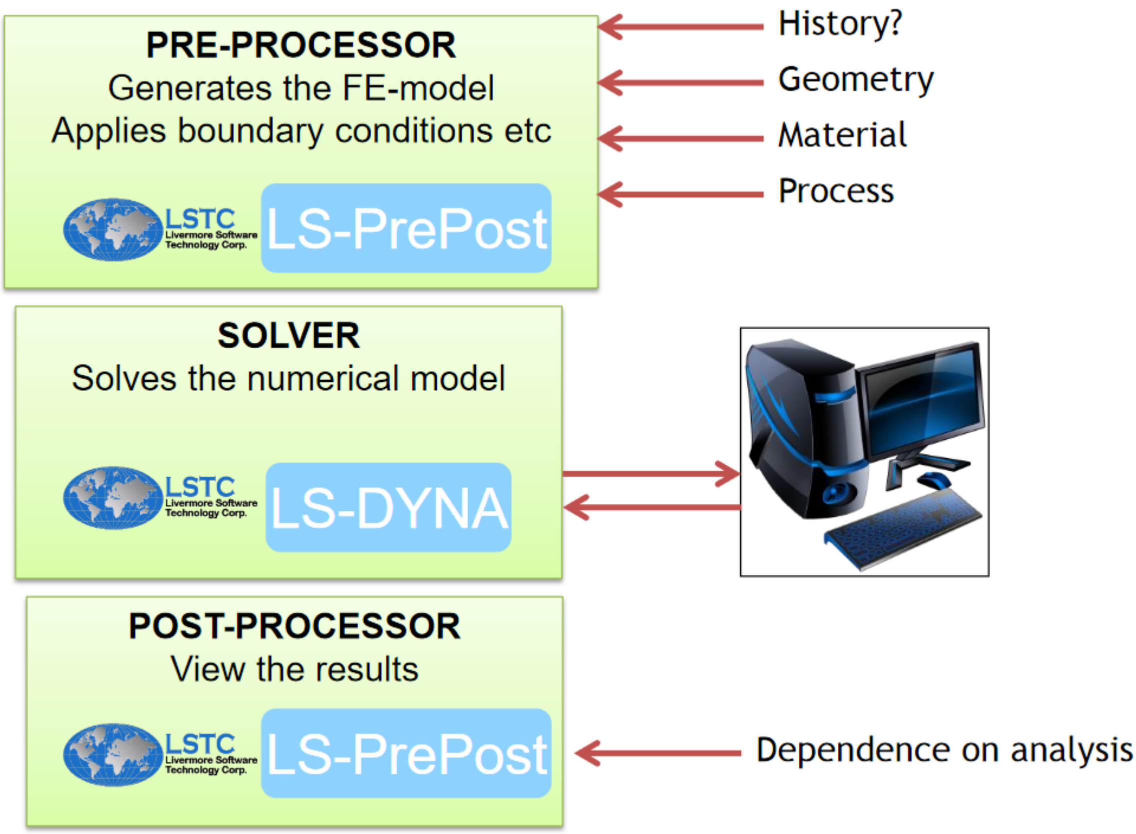

Figure 1.2: Different steps of complete LS-DYNA® finite element analysis

1.1 Aim

The aim of this tutorial is to develop a step-by-step instructions for simulation of a welding process with the FEA-program LS-PrePost®. The tutorial contain a simulation of a single-pass gas metal arc welding, but we can use the same method to simulate multi-pass welding.

2 LS-PrePost

In the preprocessing, we generate a FE model for a physical problem. There are two methods to use LS-PrePost®. The first is by means of the graphical user interface (GUI). The second is by means of keyword file. The keyword file is a text file, which builds a bridge for communication between the preprocessor and the analysis. It contains a complete description of the FE-model, like geometry, material properties, boundary condition, mesh, *etc.* This tutorial is designed to describe the user both the GUI and the keyword file approach.

2.1 Geometry

The first step in this simulation is to create and give the model the right geometrical dimensions. Geometry module is used to build different parts of the model. For complex structure we can divide the whole model into various parts and create each part using geometry module. Later all the parts can be assembled together. Here, we have a simple rectangular plate of dimension 300 × 60 × 6 mm. To create the plate geometry go to **Geometry** → **Solid** → **Box**, see Fig. 2.1.

It is also possible to import an already existing model from a CAD program.

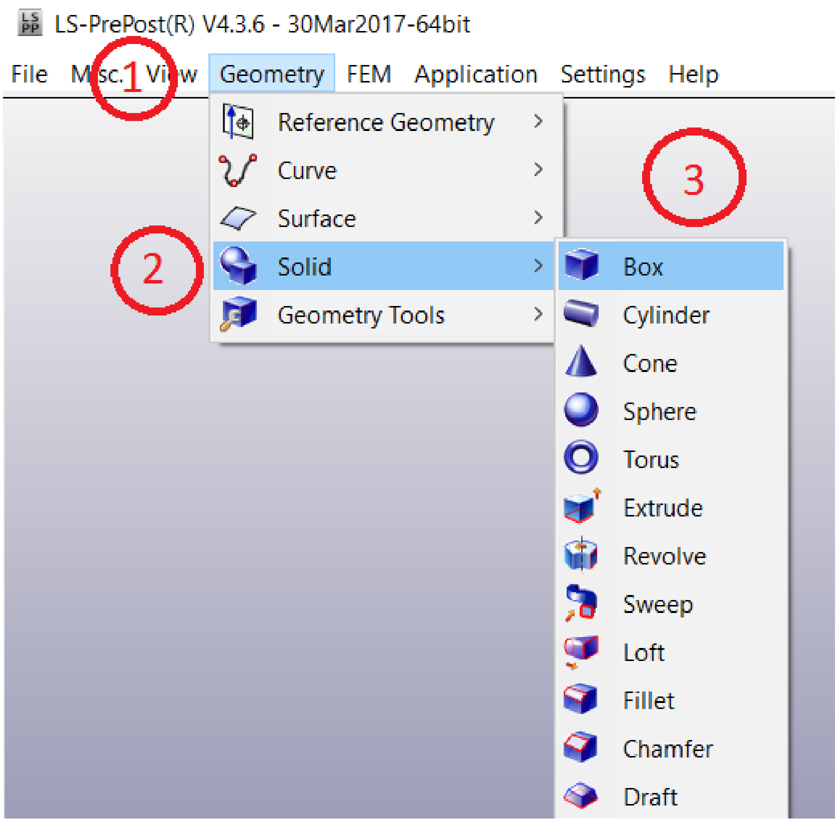

Figure 2.1: Selection of a solid box geometry

In **create Box** window choose the method parameter and enter value of the coordinate (x_1, y_1, z_1) and (x_2, y_2, z_2). At the bottom click **Apply** → **Done**, see Fig. 2.2. Instead of the menu bar one can also use the shortcut in the right toolbar, see Fig. 1.1.

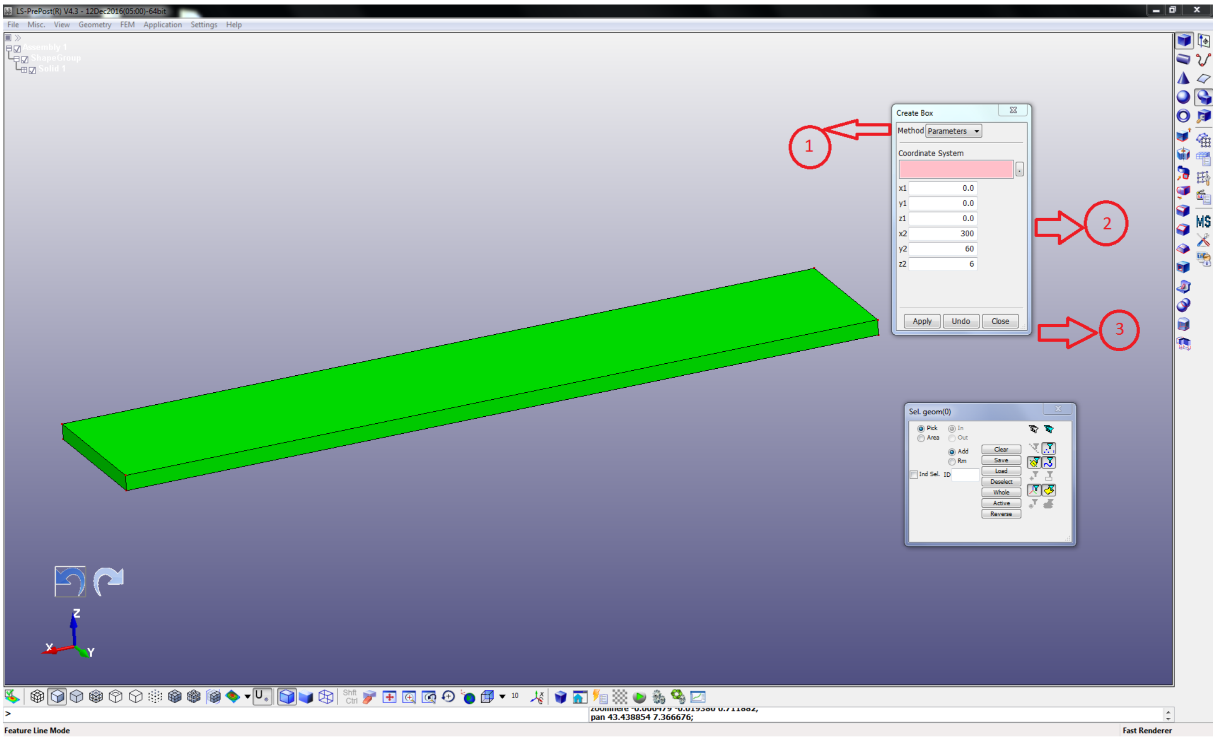

Figure 2.2: Geometry of the plate

2.2 Mesh

To create the mesh for the given plate we should use the following steps, see Fig. 2.3 and Fig. 2.4:

1. In the menu bar, click **FEM** → **Element and Mesh** → **Solid Mesher**
2. Choose Operation: **Meshing**
3. Click Volume by closed faces
4. Enter element size
5. Click **Try Meshing Automatically**
6. At the bottom click **Accept** → **Done**

This will create the mesh as shown in Fig. 2.4.

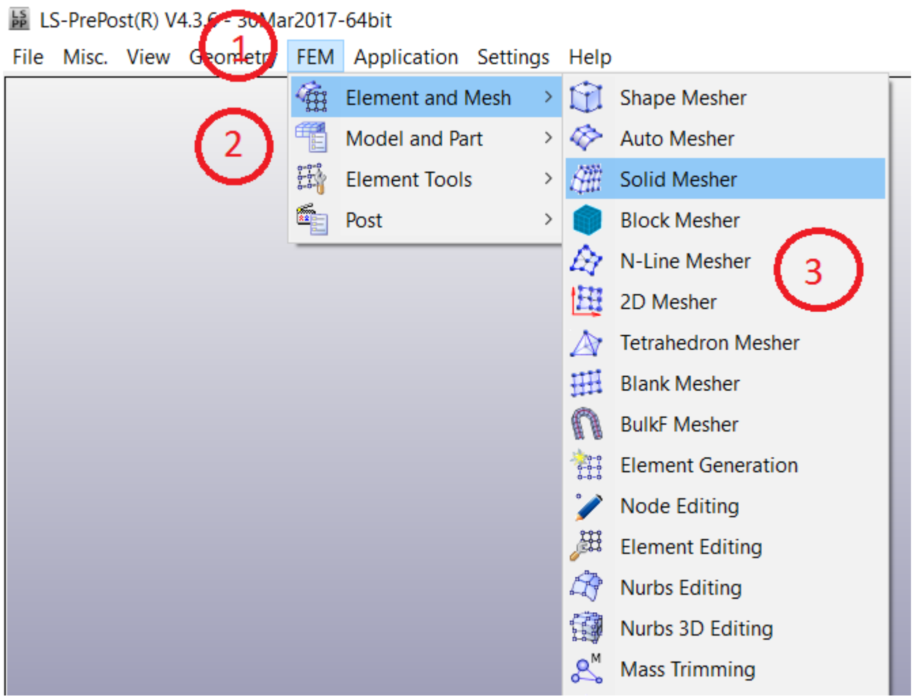

Figure 2.3: Selection of a solid mesher

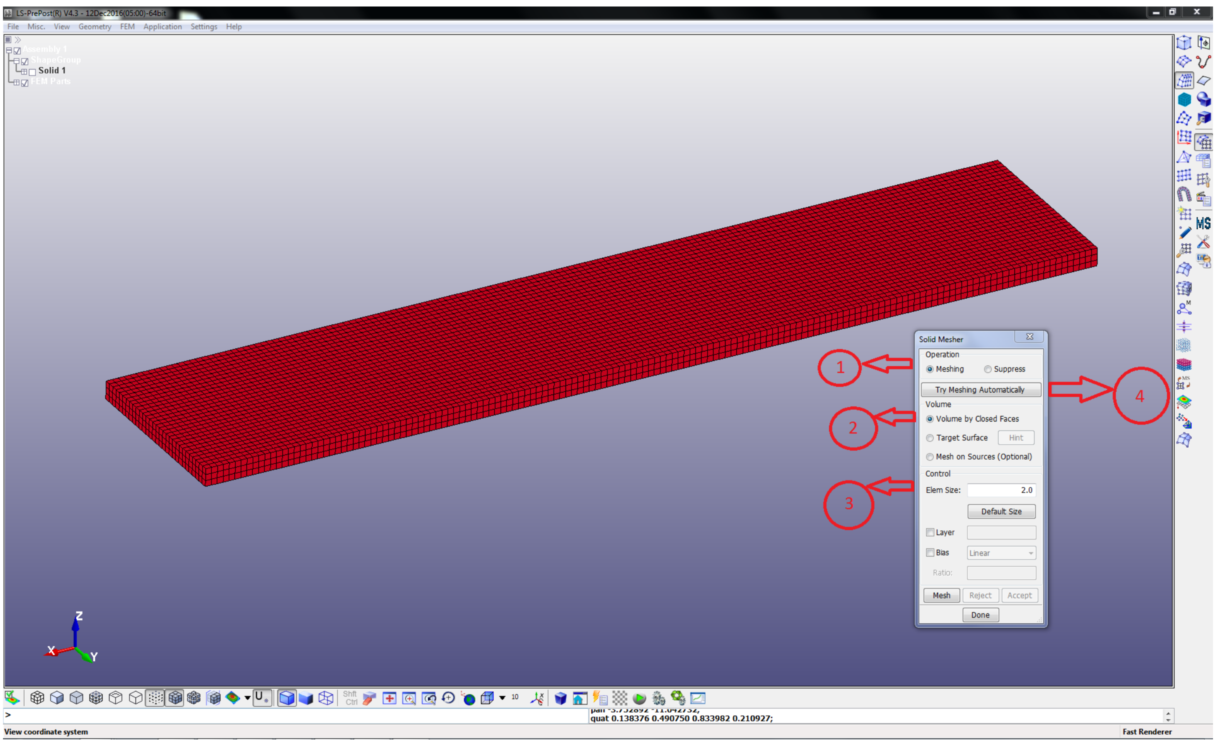

Figure 2.4: Plate mesh

We can also create mesh by using **Block Mesher**. This give us freedom to choose the number of elements in each direction manually. To create mesh by using **Block Mesher** we can use the following steps, see Fig. 2.5:

1. In the right toolbar select **Element and Mesh** → **Block Mesher**
2. Choose Operation: **Create**
3. Choose Type: **Single Block**
4. Choose eight different corner of the plate
5. Enter the number of elements in I (x-axis), J (y-axis) and K (z-axis) direction, respectively
6. At the bottom click **Create** → **Accept** → **Done**

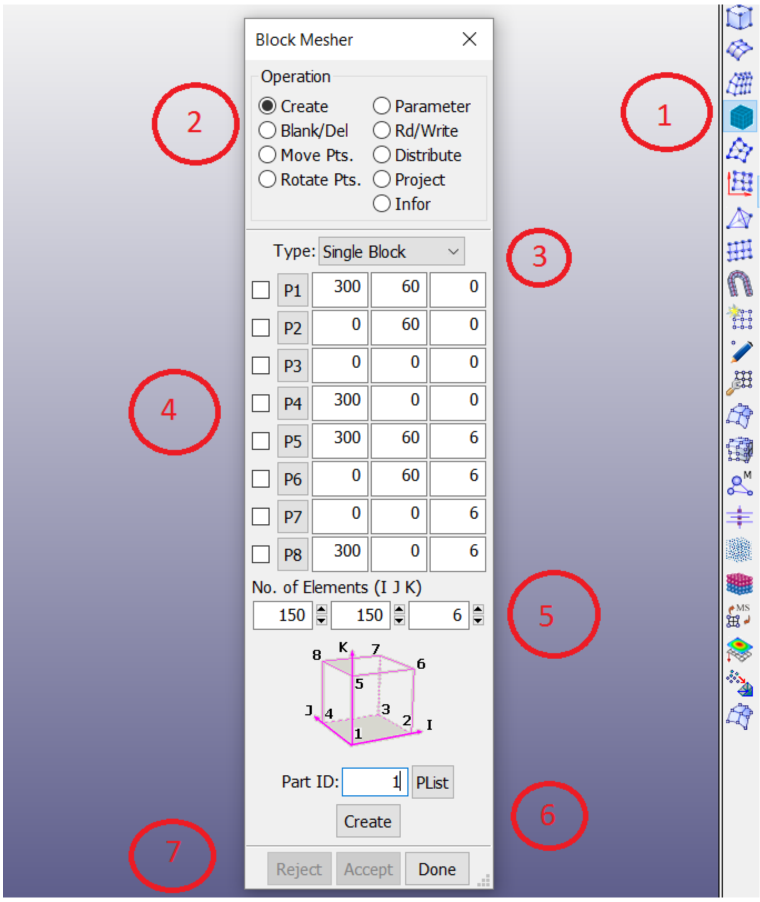

Figure 2.5: Dialog box for a Block mesher in LS-PrePost®

2.3 Welding simulation setup

For welding setup in LS-PrePost® we need to create weld and orientation paths as a beam elements. The weld path defines the path of the moving heat source represented by a local moving coordinate system. The weld path is composed of geometrical information provided from the trajectories and the time information, initial time and velocity provided from the process plan. To obtain a local coordinate system, two trajectories are required for each weld path: the weld line which describes the origin of the coordinate system and the reference line which describes its orientation.

2.3.1 Weld line

To create a weld line we need to apply the following steps, see Fig. 2.6:

1. From the LS-PrePost® menu bar, select **Geometry** → **Curve** → **Line**. A sub menu called "Create Line Segment" will pop up.
2. In the Create Line Segment dialog box choose **Parameters** as the Method, see Fig. 2.7
3. Enter the coordinate of the Start Point
4. Enter the coordinate of the End Point
5. At the bottom of the dialog box click **Apply** and **Close**

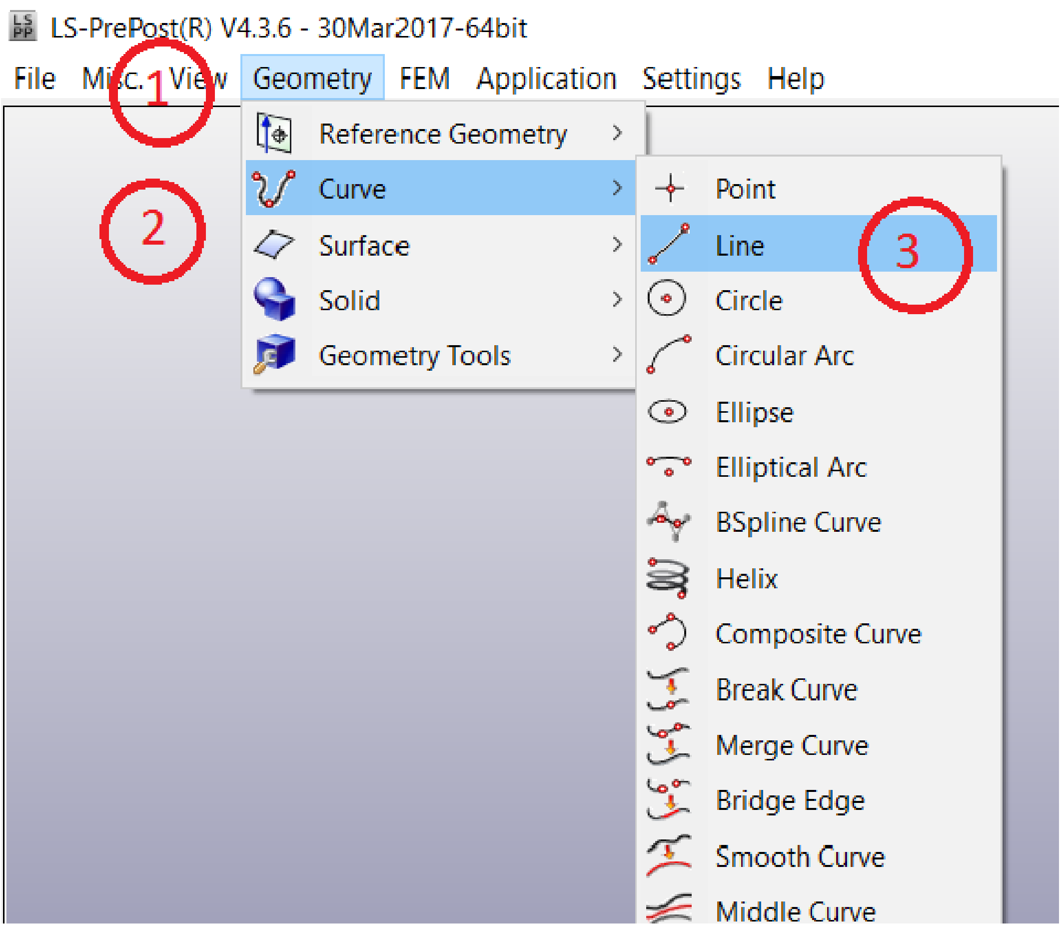

Figure 2.6: Selection of the curve toolbar to create a line

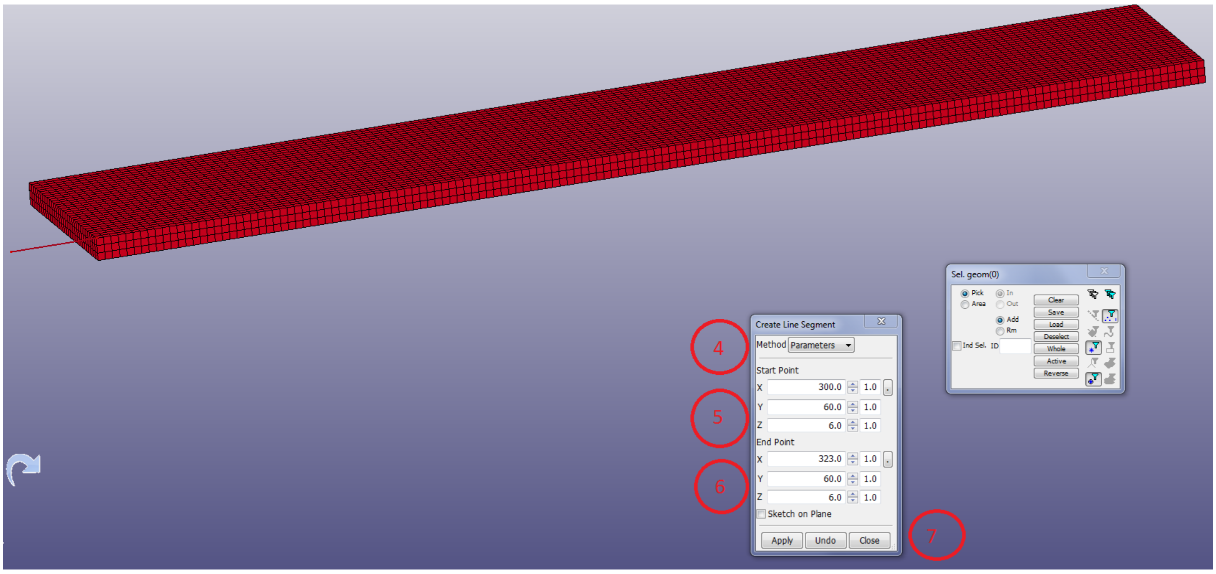

Figure 2.7: Geometry of the welding line

2.3.2 Beam element

By following the steps below we can create the beam element, see Fig. 2.8:

1. From the right toolbar, select **FEM** → **Element and Mesh** → **Element Generation**. A sub menu "Element Generation" will pop up.
2. In the Element **Element Generation** dialog box, select the option **Beam**
3. **Element ID** and **Part ID** will be automatically generated
4. Choose Beam by: **Curve**
5. Click the **Third Node** option and give the values for x, y and z
6. Choose the option **Num** and enter the value 1 in order to divide the weld line in one beam element.
7. In the next step select the line by using your cursor
8. At Bottom of the dialog box, select **Create** → **Accept** → **Done**
9. In the **Sel. genom** window, choose the options **Pick** and **add**.

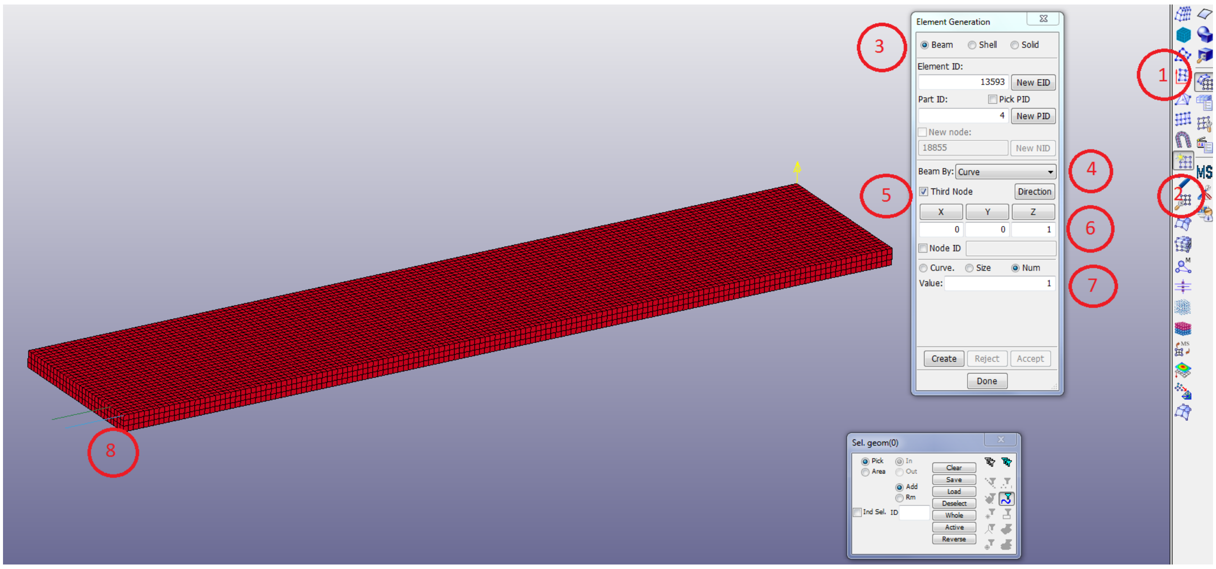

Figure 2.8: Beam element creation

2.3.3 Reference line

In LS-PrePost® we need to define a orientation path as beam element to model heat source. To create this reference line we can follow the same procedure as mentioned for welding torch, see Fig. 2.9

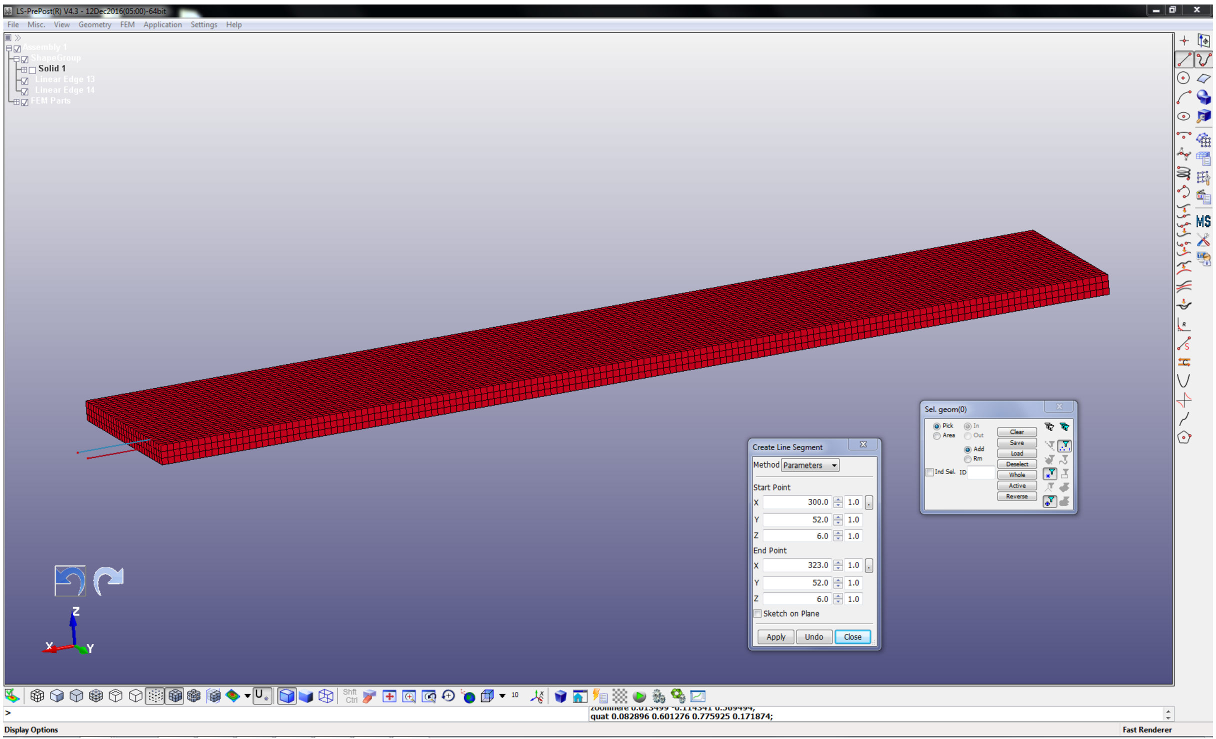

Figure 2.9: To create reference line

2.4 Weld and reference line path

To define the path of the moving heat source and its orientation, we can follow the steps below, see Fig. 2.10 und Fig. 2.11.

1. From the right toolbar, select **Model and Part** → **Create Entity**
2. From the **Entity Creation** box, select **Set Data** → ***SET_NODE**
3. Choose the option **cre** to create path for the weld line
4. Enter a **title** (This is optional).
5. From the **Sel. Nodes** dialog box, select **pick** and **add**.
6. Choose the nodes of interest along which the line will move
7. In the last step select **Apply** → **Done** from the Entity creation box

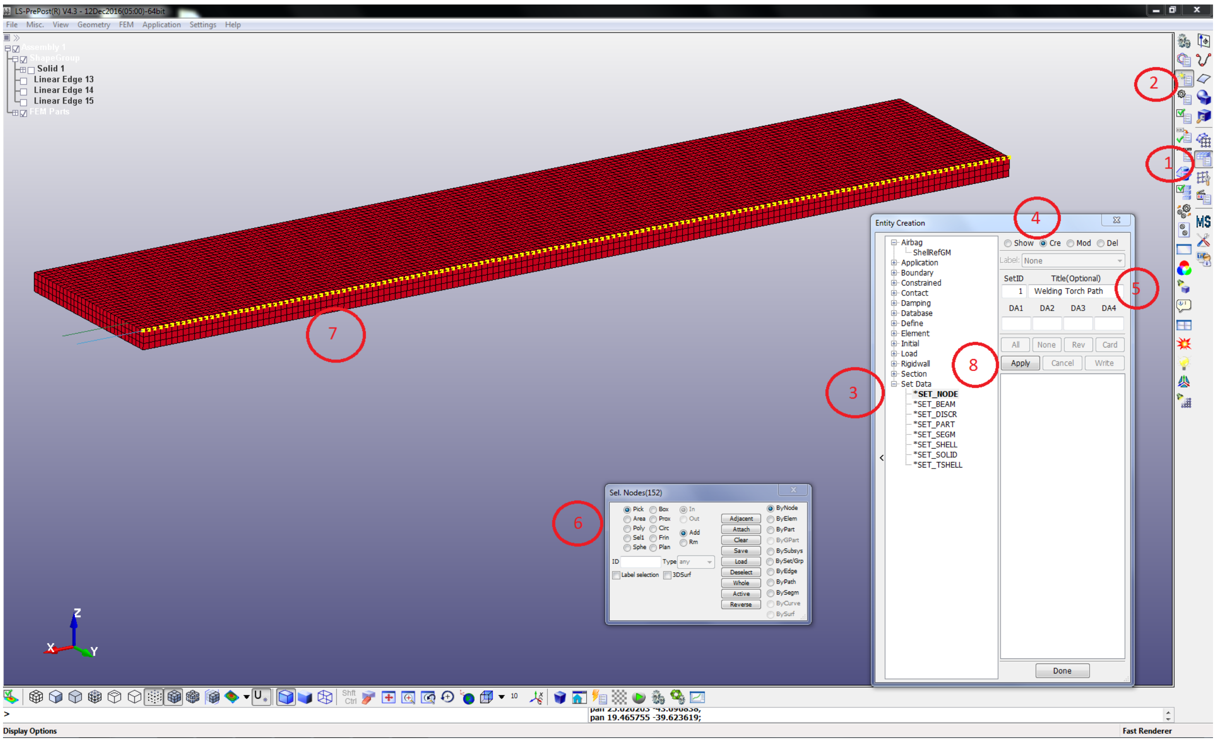

Figure 2.10: Path of the welding line

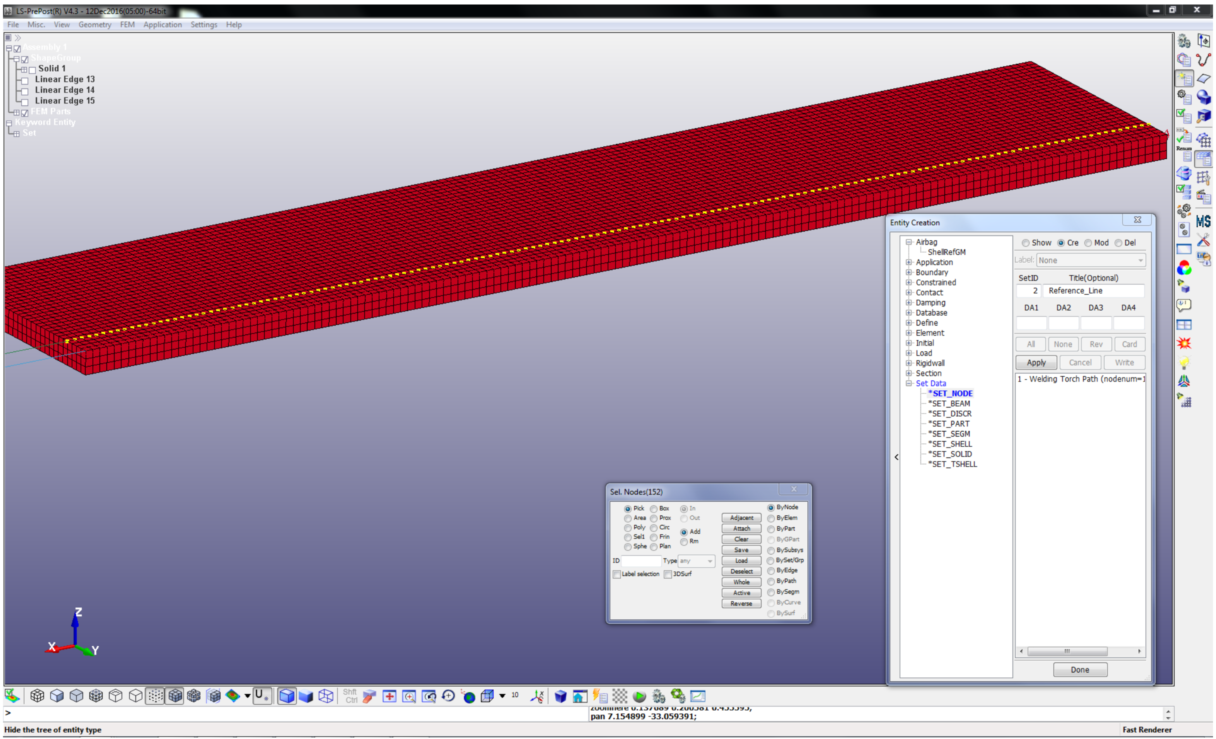

Figure 2.11: Reference line path

2.5 Material Models for Welding Simulations

The keyword ***MAT** is used to define the properties of various materials used in the model. Then, sections are created and materials are assigned to each section. For this simulation we have used the material type 270 (***MAT_CWM**). It is a thermo-elasto-plastic material model based on the VON MISES yield function with isotropic, kinematic or mixed hardening. This material model allows the parameter to be defined as a function of temperature, see Fig. 2.12. This serves as a very simplistic model for the phase change in the microstructure.

The variables, **LCEM**, **LCPR**, **LCAT**, **LCSY** and **LCHR**, in Fig. 2.12 represents the load curve for YOUNG's modulus, POISSON's ratio, thermal expansion coefficient, yield limit and hardening modulus as a function of temperature, respectively, see Fig. 2.13 - 2.17.

The material model ***MAT_UHS_STEEL** and ***MAT_PHS_BMW** are implemented for detailed studies of the material properties in the heat affected zone. This material model contains elaborate phase kinetics models and expect the input of the chemical composition and the thermos-mechanical properties of the individual phases of the microstructure.

```
$=========================================================================================
$ Mechanical material properties of the plate
$=========================================================================================
*MAT_CWM_TITLE
Plate
$#     mid        ro      lcem      lcpr      lcsy      lchr      lcat      beta
         17.85000E-9     10001     10002                          10005       1.0
$# tastart     taend   tlstart     tlend    eghost    pghost    aghost

$# t2phase   t1phase
     800.0     500.0
$=========================================================================================
```

Figure 2.12: Material property of the plate

```
$=========================================================================================
$ LCEM: E-Modul in N/mm^2
$=========================================================================================
*DEFINE_CURVE_TITLE
E-Modul(T)
$#    lcid      sidr       sfa       sfo      offa      offo    dattyp     lcint
     10001         0       1.0       1.0       0.0       0.0         0         0
$#                a1                  o1
                24.0           210000.0
               200.0           200000.0
               400.0           175000.0
               600.0           135000.0
               800.0            78000.0
              1000.0            15000.0
              1100.0             7000.0
              1200.0             3000.0
              1300.0             1000.0
              1500.0             1000.0
            100000.0             1000.0
$=========================================================================================
```

Figure 2.13: YOUNG's modulus as a function of temperature

```
$=========================================================================================
$ LCPR: Poisson's ratio
$=========================================================================================
*DEFINE_CURVE_TITLE
Poisson's ratio (T)
$#    lcid      sidr       sfa       sfo      offa      offo    dattyp     lcint
     10002         0       1.0       1.0       0.0       0.0         0         0
$#                a1                  o1
                25.0                0.3
              1400.0                0.3
              1500.0                0.3
            100000.0                0.3
$=========================================================================================
```

Figure 2.14: Poisson ratio as a function of temperature

```
$===============================================================================================
$ LCAT: Thermal expansion coeff in 1/K
$===============================================================================================
*DEFINE_CURVE_TITLE
Thermal expansion coeff (T)
$#    lcid      sidr       sfa       sfo      offa      offo    dattyp     lcint
     10005         0    1.01.00000E-6      0.0       0.0         0         0
$#                a1                  o1
                25.0               17.49
                20.0               17.49
               200.0               18.55
               400.0                19.4
               600.0                20.1
               800.0               20.85
              1000.0                21.7
              1200.0               22.35
              1300.0               22.56
              1350.0               19.66
              1500.0                 0.0
             1000000                 0.0
$===============================================================================================
```

Figure 2.15: Thermal expansion coefficient as a function of temperature

```
$===============================================================================
$ LCSY : Yield point in N/mm^2
$===============================================================================
*DEFINE_CURVE_TITLE
Streckgrenze (T)
$#    lcid      sidr       sfa       sfo      offa      offo    dattyp
     10003         0       1.0       1.0     0.000     0.000         0
$#                a1                  o1
                   0                 240
                 100                 206
                 200                 178
                 300                 160
                 400                 143
                 500                 135
                 600                 125
                 700                 110
                 800                 100
                 900                  85
                1000                  60
                1100                  30
                1200                  15
                1300                   5
                1500                   5
```

Figure 2.16: Yield limit as a function of temperature

```
$===============================================================================
$ LCHR : Hardening modul in N/mm^2
$===============================================================================
*DEFINE_CURVE_TITLE
Verfestigungsmodulus (T)
$#    lcid       sidr        sfa        sfo       offa       offo     dattyp
     10004          0        1.0        1.0      0.000      0.000          0
$#                 a1                   o1
                    0                  688
                  100                  591
                  200                  510
                  300                  459
                  400                  410
                  500                  387
                  600                  358
                  700                  315
                  800                  287
                  900                  243
                 1000                  172
                 1100                   86
                 1200                    5
```

Figure 2.17: Hardening modulus as a function of temperature

To define the thermal material property we should use the keyword ***MAT_THERMAL_CWM**. It is a thermal material model with temperature dependent properties, see Fig 2.18.

The variable, **LCHC** and **LCTC**, in fig. 2.18 represents specific heat- and thermal conductivity as function of temperature, respectively, see Fig. 2.19 and Fig. 2.20.

```
$===============================================================================
$ Thermal material properties
$===============================================================================
*MAT_THERMAL_CWM_TITLE
Plate, Trajectory and reference
$#    tmid       tro     tgrlc    tgmult
        17.85000E-9       0.0       0.0
$#    lchc      lctc   tlstart     tlend   tistart     tiend    hghost    tghost
     10006     10007
$===============================================================================
```

Figure 2.18: Thermal property

```
$===================================================================================================
$ HCLC, LCHC: Specific heat capacity in mJ/(t K) = 1e-06 J/Kg K
$===================================================================================================
*DEFINE_CURVE_TITLE
Specific heat capacity (T)
$#    lcid      sidr       sfa       sfo      offa      offo    dattyp     lcint
     10006         0       1.0   1000000       0.0       0.0         0         0
$#                a1                  o1
                00.0               430.0
               100.0               500.0
               200.0               550.0
               300.0               580.0
               400.0               610.0
               500.0               650.0
               600.0               710.0
               700.0               790.0
               800.0               865.0
               900.0               565.0
              1440.0               630.0
              2500.0               707.0
$===================================================================================================
```

Figure 2.19: Specific heat as a function of temperature

```
$===================================================================================================
$ TCLC, LCTC: Thermal conductivity mW/ mm K = W/ m K
$===================================================================================================
*DEFINE_CURVE_TITLE
Thermal Conductivity (T)
$#    lcid      sidr       sfa       sfo      offa      offo    dattyp     lcint
     10007         0       1.0       1.0       0.0       0.0         0         0
$#                a1                  o1
                00.0               0.046
               100.0               0.046
               200.0               0.045
               300.0               0.043
               400.0               0.041
               500.0               0.038
               600.0               0.035
               700.0               0.029
               800.0               0.024
              1500.0               0.032
$============================================================================================
```

Figure 2.20: Thermal conductivity as a function of temperature

2.6 Section

The keyword ***SECTION** is used to define the element formulation, integration rule, nodal thickness and cross sectional properties.

2.6.1 Plate section

After the materials have been created we need to assign these materials to the parts which were previously created. For this purpose we can use GUI as well as keyword file. To assign the plate section with corresponding material we need a section identifier (**SECID**), which must be unique and element formulation (**ELFORM**). Here, we have used the keyword ***SECTION_SOLID**, which define section properties for solid continuum and fluid elements, see Fig. 2.21.

```
$===============================================================================
$ Section - Element formulation
$===============================================================================
*SECTION_SOLID
$#   secid    elform       aet
         1         1         0
$===============================================================================
```

Figure 2.21: Section plate

2.6.2 Beam section

To define cross sectional properties for beam we should use the keyword ***SECTION_BEAM**, see Fig. 2.22.

```
$===============================================================================
$ Section Element formulation
$===============================================================================
*SECTION_BEAM
$#   secid    elform      shrf   qr/irid       cst     scoor       nsm
         2         1       1.0         2         0       0.0       0.0
$#     ts1       ts2       tt1       tt2     nsloc     ntloc
       1.0       1.0       1.0       1.0       0.0       0.0
$===============================================================================
```

Figure 2.22: Section beam

2.7 Heat source

A weld heat source model by J. GOLDAK is used by LS-DYNA to model the weld heat source. The GOLDAK weld heat source model is based on a Gaussian distribution of power density in space. An important feature of this model is that it uses a double ellipsoidal method of heat deposition such that the size and shape of the energy source can be easily changed to account for different types of welds namely, arc welding, laser and beam welding, and friction stir welding. The energy deposited in the front and rear quadrants are defined by the fractions F_f and F_r. The weld shape parameters, a, b, c_f, c_r are defined using ***BOUNDARY_THERMAL_WELD** keyword in input file, see Fig. 2.23.

```
$===================================================================================
$ Trajectory - Heatsource in Ws
$===================================================================================
*BOUNDARY_THERMAL_WELD
$#     pid      ptyp       nid     nflag        x0        y0        z0      n2id
      1001         2    128956         1       0.0       0.0       0.0    128959
$#       a         b        cf        cr     lcid          q        ff        fr
        20        20        15        30        0    5950000     1.086     0.904
$#     pid      ptyp       nid     nflag        x0        y0        z0      n2id
      1002         2    128962         1       0.0       0.0       0.0    128965
$#       a         b        cf        cr     lcid          q        ff        fr
        20        20        15        30        0    5950000     1.086     0.904
$====================================================================================
```

Figure 2.23: The GOLDAK weld heat source

2.8 Contact cable

To bring the weld line in contact with plate, we can use the keyword ***CONTACT_GUIDE_CABLE**, see Fig. 2.24.

```
$===================================================================================
$ Contact - Cabel: Define a siliding contact that guides beam element through a list of nodes
$===================================================================================
*CONTACT_GUIDED_CABLE
$#    nsid       pid      soft     ssfac      fric
         1         2         0       1.0       0.0
$#    nsid       pid      soft     ssfac      fric
         2         4         0       1.0       0.0
$====================================================================================
```

Figure 2.24: Contact

2.9 Heat source motion

The heat source movement like, translation and rotation, is defined by using the keyword ***BOUNDARY_PRESCRIBED_MOTION_RIGID**, see Fig. 2.25. .

```
$===================================================================================
$ Provide motion to the heat source
$===================================================================================
*BOUNDARY_PRESCRIBED_MOTION_RIGID
$#     pid       dof       vad      lcid        sf       vid     death     birth
         2         1         2      4001       1.0         0       0.0       0.0
$#     pid       dof       vad      lcid        sf       vid     death     birth
         3         1         2      7001       1.0         0       0.0       0.0
$#     pid       dof       vad      lcid        sf       vid     death     birth
         4         1         2      4001       1.0         0       0.0       0.0
$#     pid       dof       vad      lcid        sf       vid     death     birth
         5         1         2      7001       1.0         0       0.0       0.0
$====================================================================================
```

Figure 2.25: Motion of the weld and reference line

2.10 Weld line motion

The weld line is considered to be a rigid body and we can use the keyword ***MAT_RIGID** to define its material property, see Fig. 2.26.

```
$=========================================================================================
$ Rigid bodies properties for the welding torch: Trajectory
$=========================================================================================
*MAT_RIGID_TITLE
Welding torch
$#      mid        ro         e        pr         n    couple         m     alias
          27.85000E-9  210000.0       0.3       0.0       0.0       0.0
$#      cmo      con1      con2
        1.0         0         7
$#lco or a1        a2        a3        v1        v2        v3
        0.0       0.0       0.0       0.0       0.0       0.0
$=========================================================================================
```

Figure 2.26: Rigid body property for the weld line

2.11 Initial condition

To initialize a particular value we can use the keyword ***INITIAL**. To define initial nodal point temperature we can use the keyword ***INITIAL_TEMPERATURE_SET**. These initial temperatures are used in a thermal only analysis or a coupled thermal/structural analysis. In Fig. 2.27 we have used the numerical value 0 for **NSID**. Here, 0 represents that we have included all nodes. Instead of that we can also use particular set of nodes.

```
$=========================================================================================
$ Initial value
$=========================================================================================
*INITIAL_TEMPERATURE_SET
$#     nsid      temp       loc
          0      20.0         0
$=========================================================================================
```

Figure 2.27: Initial condition

2.12 Thermal boundary condition

For a thermal analysis we need to apply a convection boundary condition on a **SEGMENT** or **SEGMENT_SET**. For this we can use the keyword ***BOUNDARY_CONVECTION_SET** (This card is used to specify the convection coefficients to a group of element segments), see Fig. 2.28.

The variable **HLCID** represents convection heat transfer coefficient, h.

```
$========#=========#=========#=========#=========#=========#=========#=========#
$ Boundary conditions - Thermal
$========#=========#=========#=========#=========#=========#=========#=========#
*BOUNDARY_CONVECTION_SET
$#    ssid
         1
$#   hlcid     hmult     tlcid     tmult       loc
       -51       1.0         0      20.0         0
$========#=========#=========#=========#=========#=========#=========#=========#
$
$========#=========#=========#=========#=========#=========#=========#=========#
*DEFINE_CURVE_TITLE
Radiation and Convection
$#    lcid      sidr       sfa       sfo      offa      offo    dattyp
        51         0       1.0   1.0e-03     0.000     0.000         0
$#                a1                  o1
$-10000,20.0
0,20.00
100,20.00
200,20.00
300,20.00
400,20.00
500,20.00
600,20.00
700,20.00
800,20.00
1000,20.00
1250,20.00
1500,20.00
1750,20.00
2000,20.00
50000,20.00
$===============================================================================================
```

Figure 2.28: Thermal boundary condition

2.13 Curve

The keyword ***DEFINE_CURVE** is used to define trajectory and reference line travel distance, see Fig. 2.29.

```
$================================================================================================
$ Define curve for Trajectory
$===============================================================================================
*DEFINE_CURVE_TITLE
Trajectory x (t)
$#    lcid      sidr       sfa       sfo      offa      offo    dattyp     lcint
      4001         0       1.0       1.0       0.0       0.0         0         0
$#                a1                  o1
                 0.0                 0.0
                16.7              -310.0
             10000.0              -310.0
*DEFINE_CURVE_TITLE
Trajectory x (t)
$#    lcid      sidr       sfa       sfo      offa      offo    dattyp     lcint
      7001         0       1.0       1.0       0.0       0.0         0         0
$#                a1                  o1
                 0.0                0.00
                16.7             -310.00
$========#=========#=========#=========#=========#=========#=========#=========#
```

Figure 2.29: Curve definition for weld and reference line

2.14 Parameter

This option provides a way to specifying numerical values of parameter names that are referenced throughout the input file. The parameter definitions, if used, should be placed at the beginning of the input file following *KEYWORD. Here, we have assign numerical values to time step and heat source velocity, see Fig. 2.30.

```
$===============================================================================================
$ Parameter: Define the numerical values of parameter names referenced throughout the inout file
$===============================================================================================
*PARAMETER
R       dt 0.1      R        v       6.0
$===============================================================================================
```

Figure 2.30: Parameter

2.15 Part group

By using the keyword ***SET_PART_LIST** we can create part set ID, which can be used for GOLDAK double ellipsoid heat source, see Fig. 2.31.

```
$================================================================================
$ Part Groups
$================================================================================
*SET_PART_LIST_TITLE
Heat source Trajectory
$#     sid       da1       da2       da3       da4    solver
      1001       0.0       0.0       0.0       0.0MECH
$#    pid1      pid2      pid3      pid4      pid5      pid6      pid7      pid8
         1         2         3         0         0         0         0         0
*SET_PART_LIST_TITLE
Heat source Trajectory
$#     sid       da1       da2       da3       da4    solver
      1002       0.0       0.0       0.0       0.0MECH
$#    pid1      pid2      pid3      pid4      pid5      pid6      pid7      pid8
         1         4         5         0         0         0         0         0
$============================================================================
```

Figure 2.31: Part Group

2.16 Implicit analysis

Thermal problems in LS-PrePost® are solved using implicit time integration. Therefore, there is no stability condition on the thermal time step. Much larger time steps can be used for the thermal solution as opposed to the mechanical solution, which uses explicit time integration. For implicit analysis we need to use the following keywords, see Fig. 2.32 and Fig. 2.33:

***CONTROL_SOLUTION**: This card is used to specify the analysis solution procedure if thermal only or coupled thermal analysis is performed.

***CONTROL_START**: This keyword define the start time of analysis.

***CONTROL_TERMINATION**: Used to terminate the job.

***CONTROL_IMPLICIT_GENERAL**: This keyword is necessary to activate implicit analysis and define associated control parameters, like time step. This keyword is required for all implicit analyses.

```
$===============================================================================================
$ Solver
$===============================================================================================
$ Purpose: To specify the analysis solution procedure
$------x------------------x--------------------x---------------
*CONTROL_SOLUTION
$#    soln       nlq     isnan     lcint
         2         0         0       100
$-------x-----------------x--------------------x----------------
$Purpose:Define the start time of analysis
$-------x-----------------x--------------------x-----------------
*CONTROL_START
$#  BEGTIM
      0.00
$-------x-----------------x-------------------x------------------
$ Purpose: Stop the job
$--------x----------------x-------------------x------------------
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
      30.0         0     0.000     0.000     0.000
```

Figure 2.32: Solver for thermal analysis

***CONTROL_IMPLICIT_SOLUTION**: This keyword is used to specify whether a linear or nonlinear solution is desired.

***CONTROL_IMPLICIT_DYNAMICS**: The purpose of this keyword is to activate implicit dynamic analysis and define time integration constants.

***CONTROL_IMPLICIT_AUTO**: This card is used to define parameters for automatic time step control during implicit analysis.

***CONTROL_THERMAL_NONLINEAR**: By using this keyword we can set parameters for a nonlinear thermal or coupled structural/thermal analysis.

***CONTROL_THERMAL_TIMESTEP**: This card is used to define the timestep parameters for the thermal only analysis.

```
$--------x----------------x-------------------x-------------------x--------------
$ Purpose: Activate implicit analysis and is required for all implicit analysis
$--------x----------------x-------------------x-------------------x--------------
*CONTROL_IMPLICIT_GENERAL
$#  imflag    □ dt0    inform      nsbs       igs     cnstn      form    zero_v
         1      &dt         2         1         2         0         0         1
$--------x----------------x-------------------x-------------------x---------------
$ Purpose: To Specify whether a linear or nonlinear solution is desired
$--------x----------------x-------------------x-------------------x---------------
*CONTROL_IMPLICIT_SOLUTION
$#  nsolvr    ilimit    maxref     dctol     ectol     rctol     lstol    abstol
        12         1        15      0.05     100.0       0.1       0.9
$#   dnorm    diverg     istif   nlprint    nlnorm   d3itctl
         2         2                   1                   1
$#  arcctl    arcdir    arclen    arcmth    arcdmp    ARCPSI    ARCALF    ARCTIM
         0         0       0.0         1         2
$--------x----------------x------------------x--------------------x---------------
$ Purpose: Activate implicit analysis
$--------x----------------x------------------x--------------------x---------------
*CONTROL_IMPLICIT_DYNAMICS
$    IMASS     GAMMA      BETA     TDBIR    TDYDTH    TDYBUR     IRATE
         0
$--------x----------------x------------------x--------------------x-------------------
$ Purpose : Define parameters for automatic time step control during implicit analysis
$--------x----------------x------------------x--------------------x-------------------
*CONTROL_IMPLICIT_AUTO
$#   IAUTO    ITEOPT    ITEWIN     DTMIN     DTMAX     DTEXP     KFAIL    KCYCLE
         1        16        10     0.001      -111
$--------x----------------x------------------x--------------------x----------------------
$ Purpose: Set parameters for a nonlinear thermal or coupled structural/thermal analysis
$--------x----------------x------------------x--------------------x----------------------
*CONTROL_THERMAL_NONLINEAR
$#  refmax       tol       dcp    lumpbc    thlstl    nlthpr    phchpn
        60       0.0       0.5         0       0.1         1       0.0
$--------x----------------x------------------x--------------------x-------------------------------------
$ Purpose: Set options for the thermal solution in a thermal only or coupled structural thermal analysis
$--------x----------------x------------------x--------------------x-------------------------------------
*CONTROL_THERMAL_SOLVER
$#   atype     ptype    solver     cgtol       gpt    eqheat     fwork       sbc
         1         1        11    1.0E-4         8       1.0       1.0       0.0
$#  msglvl    maxitr    abstol    reltol     omega    unused    unused       tsf
         0        50         0         0       1.0         0         0       1.0
$--------x----------------x------------------x--------------------x-------------------------------------
$ Purpose: Set time step control for the thermal solution
$--------x----------------x------------------x--------------------x-------------------------------------
*CONTROL_THERMAL_TIMESTEP
$#      ts       tip       its      tmin      tmax     dtemp      tscp      lcts
         0      1.00       &dt     0.000     0.000      5000      0.50       112
$=======================================================================================
```

Figure 2.33: Solver for implicit thermal analysis

2.17 Output file

The keyword ***DATABASE** with a combination of options is used for controlling the output of **ASCII** databases and binary files output. With this keyword the frequency of writing the various databases can be determined.

The keyword ***DATABASE_BINARY** is necessary to obtain output files containing results information, see Fig. 2.34. The keyword ***DATABASE_TPRINT** gives thermal ouput from a coupled structural/thermal or thermal only analysis.

The keyword ***DATABASE_BINARY** is used for binary output option and the attribute D3PLOT represent the database for entire model. If ***DATABASE_BINARY_D3PLOT** is not specified in the keyword file then the output interval for d3plot is automatically set to 1/20$^{\text{th}}$ the termination time.

```
$===================================================================================
$ Postprocessing
$===================================================================================
*DATABASE_TPRINT
$#      dt    BINARY      LCUR     IOOPT   OPTION1   OPTION2   OPTION3   OPTION4
      &dt         1         0         0         0
*DATABASE_BINARY_D3PLOT
$#      dt      lcdt      beam     npltc    psetid
      &dt         0         0         0         0
$#   ioopt
         0
$===================================================================================
```

Figure 2.34: Binary output file

By using the keyword ***DATABASE_EXTENT_BINARY** we may control to some extent the content of binary output databases.

```
$===================================================================================
$ Postprocessing: Structural mechanics
$===================================================================================
*DATABASE_EXTENT_BINARY
$#   neiph     neips    maxint    strflg    sigflg    epsflg    rltflg    engflg
        11        11         3         1         1         1         1         1
$#  cmpflg    ieverp    beamip     dcomp      shge     stssz    n3thdt   ialemat
         0         0         0         1         1         1         2         1
$# nintsld   pkp_sen      sclp    unused     msscl     therm    intout    nodout
         0         0  1.000000         0         0      2ALL       ALL
$#    dtdt    resplt
         1         1
```

Figure 2.35: Extended binary output file

3 Postprocessing

The results generated from the analysis can be enormous so it requires additional processing which is termed postprocessing. LS-PrePost® GUI is used for postprocessor. It supports the latest Open-GL standards to provide fast rendering for fringe plots and animation results. It also handles the **ASCII** output data and links it to the input files and animations.

LS-PrePost® also include features like animations, contour plots, X-Y graphs, overlay plots, vector plots, ASCII plotting, mesh manipulation, *etc,*.

3.1 Animation interface

The Animation controls are displayed when d3plots are loaded. If the animation interface is closed, it can be restored by clicking the **Anim** render button, see Fig. 3.1.

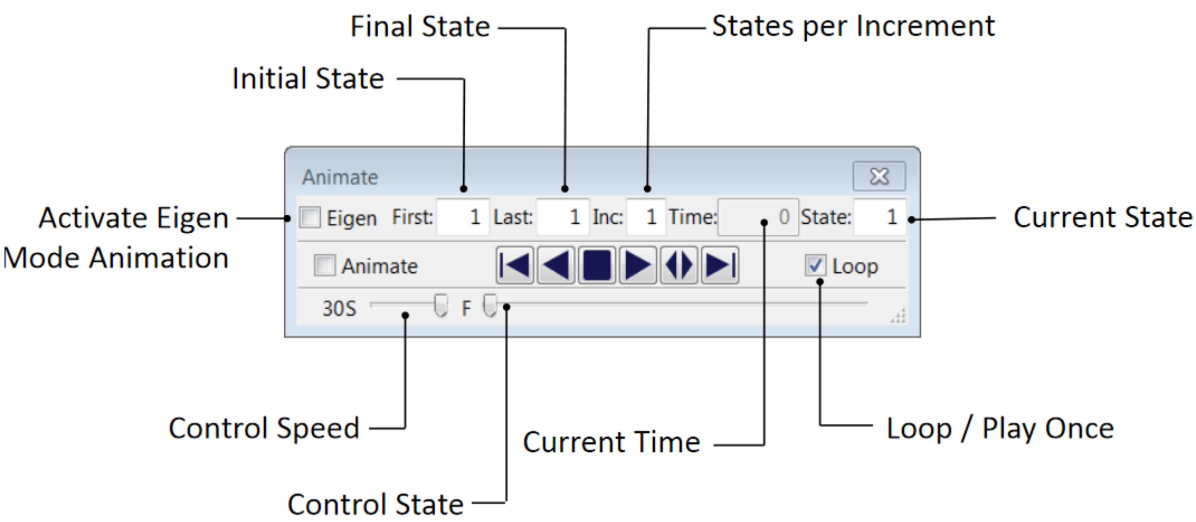

Figure 3.1: Animation interface

The results generated from the analysis can be viewed in LS-PrePost®, see Fig. 3.5:

1. From the right toolbar, select the **Fringe component** option
2. In the **Fringe component** dialog box, choose the option **Misc**
3. Choose the variable of your interest, here it is temperature
4. Use the animation interface, see Fig. 3.1, to animate the temperature field

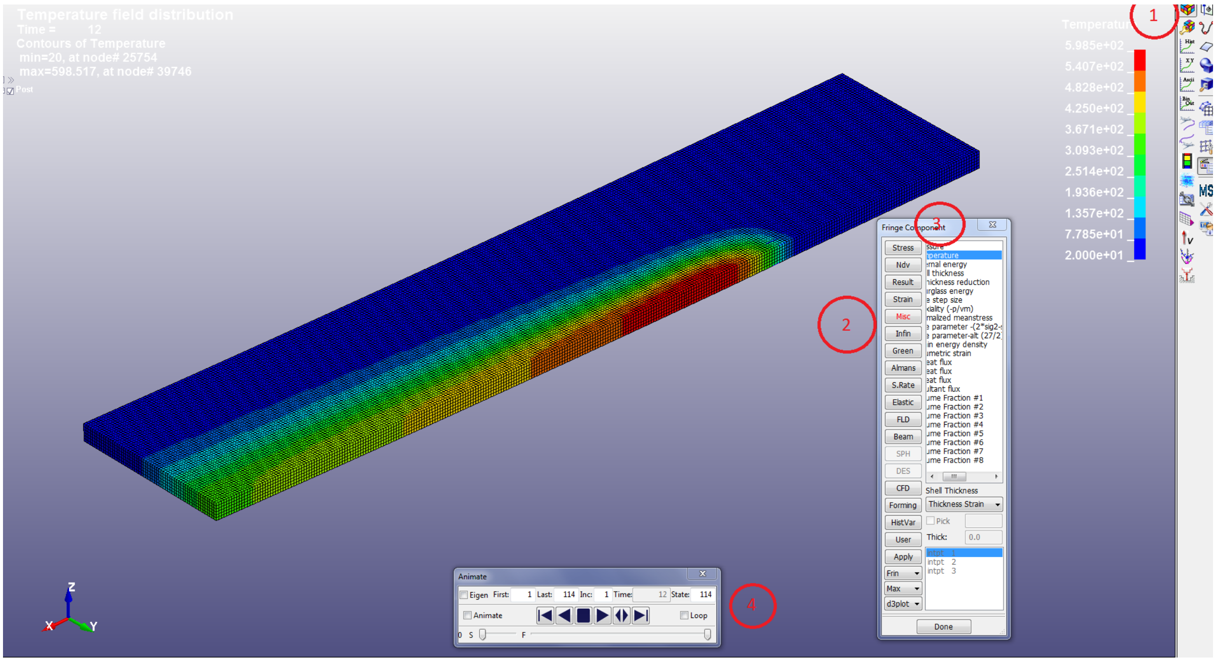

Figure 3.2: Animation interface

3.2 Plot

The temperature vs time plot can be created as shown in Fig. 3.4 - 3.5:

1. In the menu bar, select **File** → **Open** → **LS – DYNA Binary Plot**, see Fig. 3.3
2. From the right toolbar, click the **ASCII** option
3. Choose the option **tprint*** for temperature data
4. In the next step load the computed temperature data by clicking the option **Load**
5. Select the node at which you want to know the temperature distribution
6. Click the variable of interest, here temperature
7. At the bottom of the dialog box click the option **Plot** and this will generate the temperature field vs time graph as shown inn Fig. 3.5.

Here, not all the modules of postprocessing are explained. One can explore different modules and try to generate plots for different parameters and get meaningful results out of them.

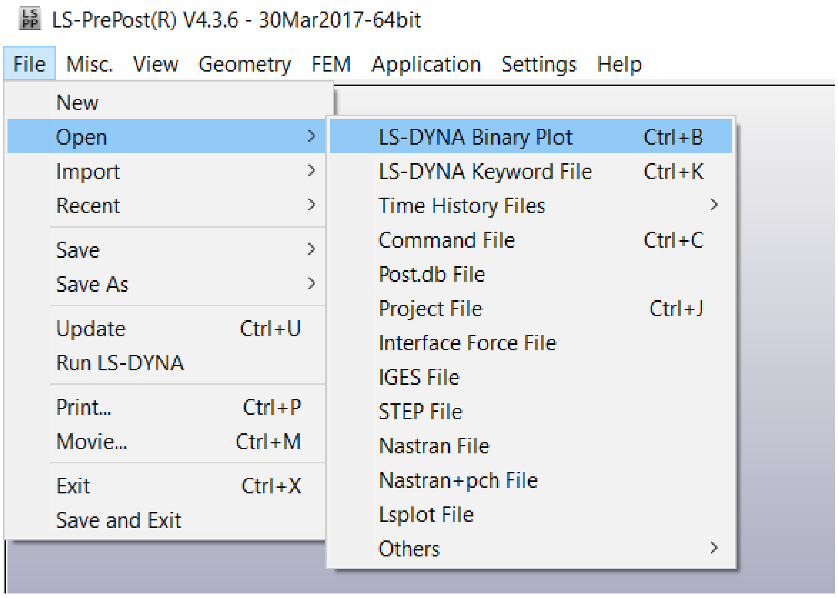

Figure 3.3: Sub menu to open the binary file

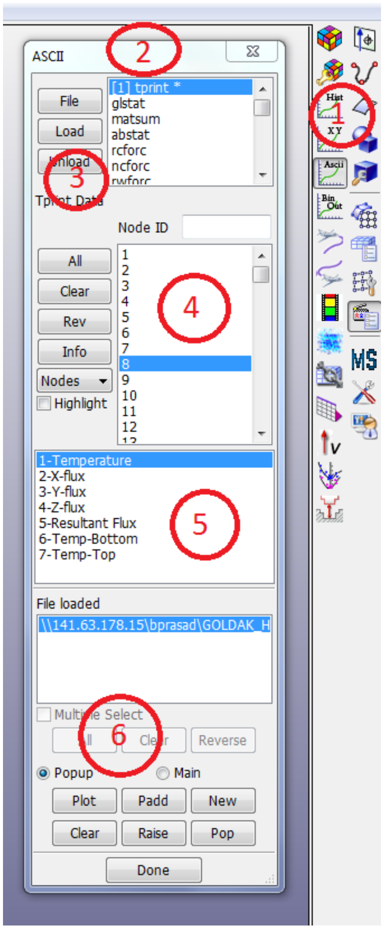

Figure 3.4: ASCII dialog box

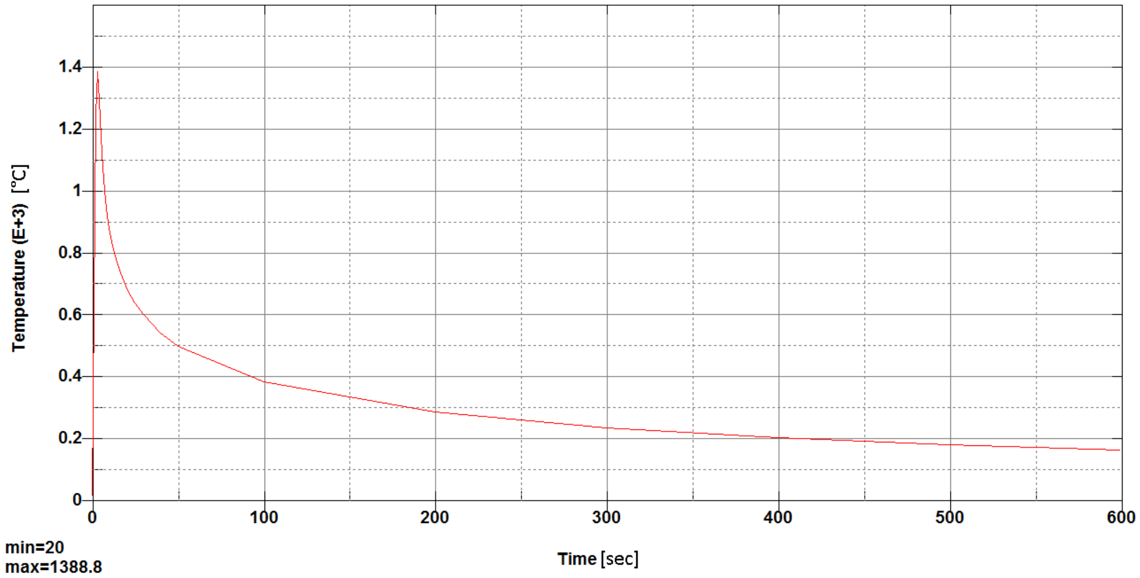

Figure 3.5: Temperature vs time graph