MaPoss System User Manual

General Equipment Department of Engineering Design Institute
Tsinghua University Aerospace Institute of Computational Dynamics

(Translated from Chinese by Zachary Burell via Google Translate, Sept 17')

Table of Contents

Chapter 1 Introduction to MaPoss

The impact collision problem is a geometrical, material and boundary conditions that are non-linear multi-physics field coupling problems involving high speed, high pressure, high temperature, phase change and chemical reaction, gas, liquid and solid and other substances between the coupling or even mixed, material will not only occur serious distortion and broken, but also melting or even vaporization. The Lagrangian method can easily track the boundaries of the material and the introduction of the material model associated with the deformation history, but for the problem of large deformation due to the serious distortion of the grid a numerical solution is difficult, and it is difficult to effectively simulate the material crushing, melting and vaporization and other acts. The Euler method does not ask for grid distortion problem, suitable for the analysis of large deformation and flow problems, but not easy to track the material interface, and non-linear convection items will lead to numerical difficulty in solving problems.

By the particle grid method (particle in cell, PIC) inspired material point method (material point method, MPM) [1, 2] using Lagrange and Euler dual description (ALE), the object is separated into a group of particles in the space grid movement.

Quality

The point carries all the material information, such as quality, speed, strain and stress, and can easily track the material interface and A material model related to deformation history. The particles move in the space grid, and the equations of motion are solved on the spatial grid. The problem of grid distortion is suitable for the analysis of large deformation and flow problems. (The) Material point method fully absorbs the Lagrange law and Euler. The advantages of the method are one of the most promising methods for ultra-high speed collisions and numerical analysis of explosions.

MPM3D® is the Computational Dynamics Laboratory of Tsinghua University (http://www.comdyn.cn) Professor of the research group since 2004 developed the

impact of the explosion three-dimensional material point method numerical simulation software (computer software copyright board Symbol for 2009SRBJ4761), can be used to simulate ultra-high-speed collision, shock, penetration and explosion problems. In order to facilitate further expansion, change, and maintenance, in summer of 2007, we rewrote the original FORTRAN 90 version of MPM3D using C++ MPM3D ®, of the MPM3D? Stop

Only development, and as a monograph (Zhang Xiong, Lian Yanping, Liu Yan, etc. "Material Point Law", Tsinghua University Press, 2013),Supporting open source code, can be downloaded from the website http://www.mpm3d.com.

The MaPoSS system is the result of the joint effort of the engineering department of the General Assembly Department and the Laboratory of Computational Dynamics of the School of Aeronautics and Astronautics, Tsinghua University, Developed, with MPM3D® as the core solver of the impact explosion numerical simulation system, which is based on Qt, VTK, CMake and

C ++ development, with excellent cross-platform features, can run on Windows, Linux, Mac OS and other operating systems.MaPoSS is a job-oriented numerical simulation system. Jobs have the following attributes:

1. **Folder**; Job's data file storage path, model data file and all output data The files are stored in that In the path.
2. **JobName**; The name of the job, uniquely marked for the job.
3. **Header;** Job title, the main user marked the job. The value of this property will be saved in the model data file In the Header element of the Header element.
4.**Description;** A brief description of the job, such as the main physical phenomena of the problem, references, and so on. This attribute
The value is stored in the description subelement of the Header element in the model data file.
5. **Job Units**; The unit system used by Units Job. It has three components, respectively, the work of the time to define the time unit, Length units and mass units, which are stored in the model data file Unit sub-elements of the time, length and mass attributes.
6. **Model dataset**; The dataset is a collection of model data for the job, including particle data, background grid data,
Solution and result output strategy and other model data, the user can use MaPoSS graphical interface to the model data set to edit all model data. The model dataset inherits naming conventions from JobName, for example, model

data for a job named Rings, (the name shown in the Job Browser panel) is also named Rings.

7. **Model data file;** model data file is to store all the attributes of the disk file, the user can use The graphical interface of MaPoSS saves the attributes of the Job into the model data file, or the attributes of the Job from the model data

Read into the MaPoSS system. Therefore, the model data file is actually a file corresponding to the job, which can be read by MPM3D® for simulation. The model data file is named after JobName. For example, named Rings Job, whose model data file is Rings.xmp, stored in the directory specified by the property Folder.

The model data file is in standard XML format and can be used with any XML editor or text editor to edit. MaPoSS has built-in QXMLEdit and has modified the editor so that it can be loaded MPM3D® material library, and can call MPM3D ® its current editing data files for simulation.

MaPoSS allows you to create or load multiple jobs, which are listed in the Job Browser panel. Users can use the mouse or keyboard to select a job in the Job Browser panel and edit all the attributes of this job, Save this Job (save to file /Folder/JobName.xmp), you can also directly edit the model data file for this job

(/Folder/JobName.xmp) and call MPM3D® to calculate the simulation for this Job.

MaPoSS system with pre-processing (Models menu), simulation calculation and monitoring (Simulation menu) two functions, Postprocessing is done using ParaView. ParaView is made by Kitware Corporation, Los Alamos National Laboratory, An open source multi-platform data analysis and visualization developed jointly by the Sandia National Laboratory and CSimSoft Inc. Application, can be ultra-large-scale data analysis and processing, can run on the desktop, can also run on the parallel computing platform

on.

Chapter II Pre-Processing System

PeneBlast system pre-processing functions mainly through the menu bar, task list area and data set attribute area to complete. The user can interactively edit the attributes and model datasets of the Job, such as creating geometric entities with regular shapes and user self

Define geometric entities, set background grids and various types of solver parameters, and export Job attributes and model datasets to model numbers According to the file (extension xmp), read for MPM3D® to solve. Pre-processing system interface includes five sections, namely the menu bar, task list area (Job Browser), data set

Job Properties, GUI real-time display area, and real-time monitoring area. PeneBlast
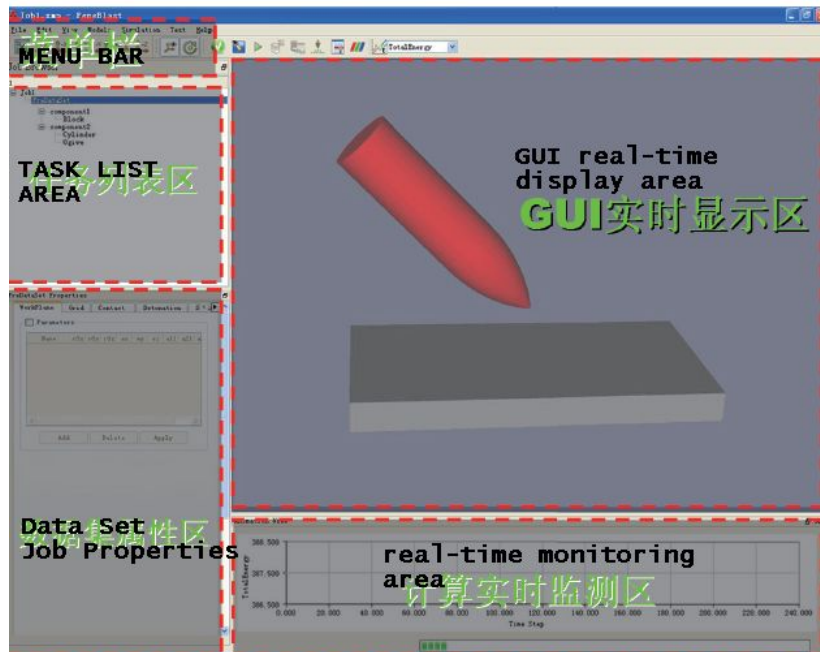
pre-processing system interface tour

Figure 2.1 shows the figure.

2.1 Pre-processing system functions and process overview

2.1.1 Pretreatment System Function Description

Each submenu in the menu bar contains all the pre-processing action buttons, which can be edited interactively by these menus job attributes and model data sets, to complete the pre-processing settings and operation. The following describes each drop-down menu function:

- **File**
  File submenu (Figure 2.2) can complete the file, task-related operations.

- **New Job**

  Create a new task to solve the task, while the task list will be automatically numbered new tasks, the user can click on the Job
  The Job Name edit box in Properties changes the name.

- **Open Job**

  Open Job function, unlike the Load Job, only adds the specified job to the Job Browser list, The PeneBlast system does not read the actual content of the xmp file. After selecting the job, you can edit its xmp text
  Pieces, check the xmp file format, call MPM3DPP to calculate, view the log file.

- **Load Job**

  Import the task, according to the pop-up file dialog box (Figure 3.1), import the specified path under a pre-processing model data text (.xmp file), the pre-processing system will load the corresponding parameters, while the model displayed in the GUI real-time display area in. The file holds all the information entered in the pre-processing interface, which is processed by the "Save Job" menu of the preprocessing system generated by the user can also use the text editor to generate their own, you can import the pre-processing interface for users to check model and reset.

- **Save Job**

  Save the current model's status to the current path (the path shown in the Folder box in the Job properties panel path), specify the name (the name shown in the JobName box in the Job properties panel) in the xmp format In the file.

- **Save Job As**

  Save the task, according to the pop-up dialog box (Figure 2.4) *(translator note: figure omitted typical "save-as" dialog))*the current task to specify the name (* .mlb) to the specified In the path.

- **Save Screenshot**

  Screenshot function, according to the pop-up dialog box (Figure 2.5), the GUI real-time display area image copied to the specified path.

- **Exit**

  Exit the PeneBlast system.

- **Edit**

  The Edit submenu (Figure 2.6) allows you to delete the Job function and make the relevant settings.

- **Delete**

  Deletes the currently selected job

- **Delete All**

  Delete all jobs in the task list area

- **Setting**

  Set the working path and the text editor, ParaView executable file, text comparator path, click "Apply" Button to complete the setting, as shown in Figure 2.7.
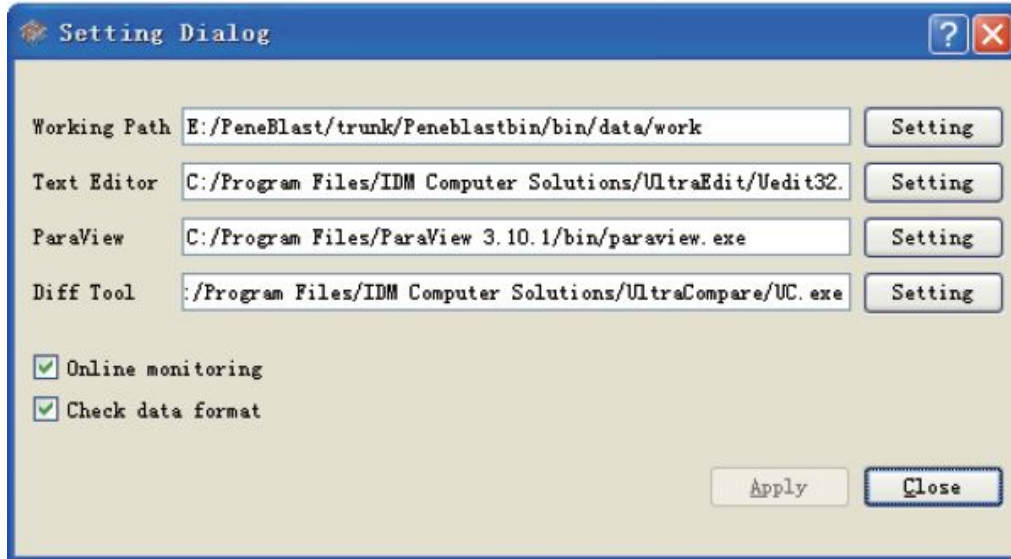
图 2.7: Setting 对话框

- **View**

  The View submenu option (see Figure 2.8) performs a variety of view operations on the GUI real-time display area.
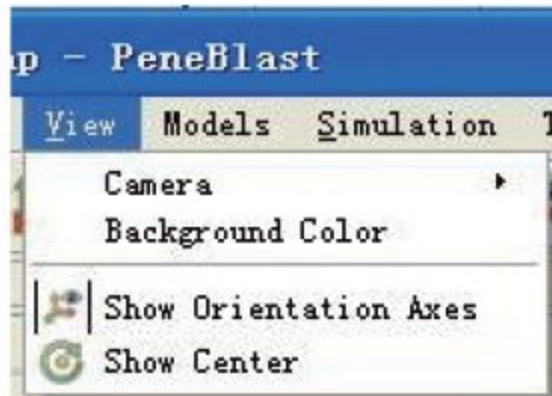
图 2.8: View 菜单

- **Camera**

  Reset - Center all live to the GUI real-time display area

+ X, -X, + Y, -Y, + Z, -Z- Set the direction of the observer to X-axis positive, X-axis negative, Y-axis forward, Y-axis negative, Z-axis positive, Z-axis negative.

● **Show Orientation Axes**

Whether the global coordinate frame is displayed in the GUI real-time display area

● **Show Center**

Whether the global origin is displayed in the GUI real-time display area

● **Models**

Models submenu (Figure 2.9) can complete the material definition, create a new component, create a new geometry with regular shape (Block, Cylinder, Ogive, Sphere) and user-defined geometric entities (User-Defined).
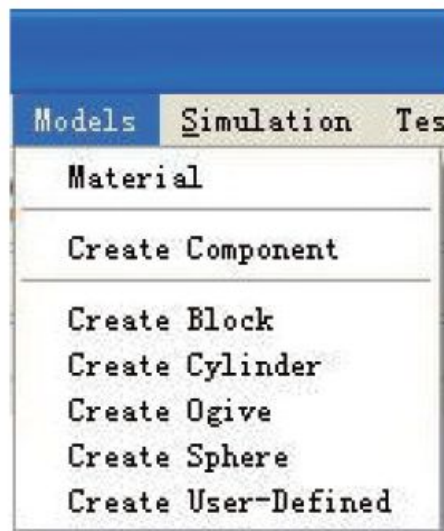
图 2.9: Models 菜单

● **Material**

Define the material for the current task. For details, see Section 2.3.

● **Create Component**

New components, the current task of the new components will be automatically numbered, the user can double-click the task list area on the component name
To modify the component name.

● **Create Block**

Create a new block. For details, see Section 2.5.1.

● **Create Cylinder**

New generalized Cylinder (regular rotation), see Section 2.5.2 for specific operations.

- **Create Ogive**

  Create a new Ogive, see Section 2.5.3 for specific operations.

- **Create Sphere**

  Create a new Sphere, see Section 2.5.4 for specific operations.

- **Create User-Defined**

  The new user imports the 3D solid model (stl format). For details, see Section 2.5.5.

- **Create User-Defined**

  Create a new user-defined geometry entity. For details, see Section 2.5.6.

Simulation

The Simulation submenu (Figure 2.10) completes the check, edit, and start of the current xmp file format and other functions.
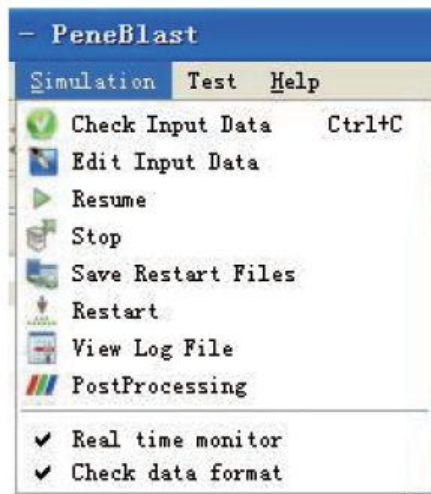
图 2.10: Simulation 菜单

- **Check Input Data**

  When the task the format is wrong before the xmp file generated by the format check, and give the test results - pop-up dialog box (Figure 3.2), given the specific error message; format is correct pop-up dialog box (Figure 3.3) prompt format check Check through.

- **Edit Input Data**

  Call the text PeneBlast system comes with the editor QXmlEdit on the former task generated by the xmp file Series, modify and other operations, as shown in Figure 2.13.

14
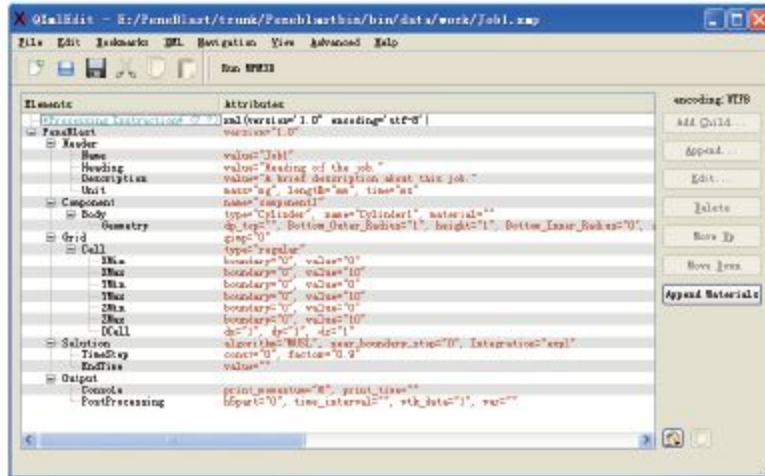
图 2.13: QXmlEdit 界面中打开当前任务文件

- **Start**
  Call the MPM3DPP program to start calculating the current task.

- **Real time monitor**
  Whether real-time display of the calculation process is enabled in the GUI real-time display area.

Test

The Test submenu (Figure 2.14) is used to complete the study file in the standard study database TestDatabase (xmp Format) of the calculation test function. Users click the Test button that pops up MPMTest dialog box (Figure 2.15), the user can As needed to detect all or part of the xmp file under the specified path to observe the results of the latest MPM3D calculation program And the standard study database TestDatabase in the results of the document is consistent.

图 2.14: Test 菜单

Test directory interface (fig 2.15), specifies the need to detect the xmp file sitting in the location (path); Output directory Specify the result output path to calculate the test; the "Search" button is used to search all the xmp files under the Test directory, And the list is displayed; the "Select" button is used to select all or all of the xml files in the list box;

Click the "Test" button to start the calculation of the selected xmp file test; click "Stop" button to stop the test; point Click the "Close" button to exit the calculation test and close the MPMTest interface dialog box. Among them, the list box "Status" gives the results of each study to calculate the test and the standard study database Test-Database in the results of the file comparison, if the two results are consistent, otherwise displayed.
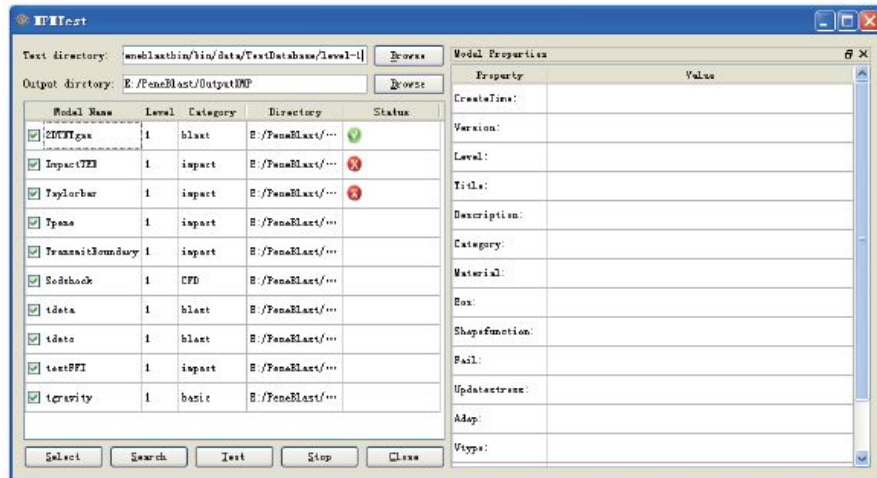
图 2.15: MPMTest 界面对话框

- **About**
  Given the PeneBlast system profile, R & D team, release log and other information.
- **PeneBlast User's Manual**
  View the PeneBlast User Manual
- **MPM3D User's Manual**
  View the MPM3D User Manual
- **MPM3D Theory Manual**
  View the MPM3D theory manual
- **MPM3D Example Manual**
  See the MPM3D study manual

2.1.2 Preprocessing Process Overview

The ultimate task of a typical pre-processing process is to generate a mass-value simulation program MPM3DPP read the solution of the data file (extension xmp), therefore, the pre-processing work mainly includes the following steps:

1. New solution task;
2. Define material - New material model, or import material data directly from existing

libraries;

3. New components in the pre-processed data set (PreDataSet);

4. Create a new geometry under the various components - the new conventional geometric entities (Block, Sphere, Cylinder, Ogive) And it is in the form of a discrete form of material points; or directly from the third party CAE software into the built a few what information is scattered.

5. Set the attributes of the pre-processed data set - set the work plane, background grid information, contact control, definition
Burst point (face), solution options, output settings.

6. Save the specified job, output the pre-processing task file - generate the mass point for the simulation program MPM3DPP read
Solve the data file (extension xmp).

7. Calculate the operation, call the material point value simulation program MPM3DPP, start calculating the new task (file extension
xmp).

2.2 The 'New Solution' Task

Function: the new task to solve the task, specify the task file (*. Xmp) output path to solve the task named 'JobName', A brief description of the tasks is made, and the unit system of the three basic dimensions (quality, length, time) of the pre-processing set is set.

The new solution task is as follows:

1. Click "New Job" on the File menu to automatically create a Job in the Job List area
And its corresponding PreDataSet;

2. Click to activate the current task of the task list area, in the bottom of the Job Properties column will pop up the task settings panel (as shown in Figure 2.17);

3. Select the storage path "Folder" of the new task. This field is required. The default folder is "installation path / data / work" The default path can be changed by clicking the "Browser" button;

4. Specify the task name "JobName", the user changes the default name of the task will be displayed in the task list synchronization, This field is required, the file name of the output xmp file is "JobName.xmp";

5. Add the task title "Heading" (which lists the attributes such as the classification of the problem) and the task description "Description"
The problem is briefly described, the content will be written to the xmp file comments); "Heading" and "Description" also can not fill
6. Select the unit used in this task - by selecting the "Mass" "Length" "Time" checkbox Bit, to specify the unit system used. Among them, the mass mass of the optional units for the mg, g, kg, length. The optional units of the Length dimension are mm, cm, m, and the optional units of the Time Time dimension are s, ms, s; The unit of acceptance is mg-mm-ms.

图 2.17: Job properties 设置面板

The JobBrowser panel shows the only corresponding data set for the job and the components and geometries contained in the dataset type. For each Job, its name, the selected unit system, xmp file name and storage path, etc. can be modified First select the mouse with the need to modify the task list area of the Job, by modifying the following Job Properties column settings board corresponding options, you can complete the corresponding edit box parameters of the changes, update. At the same time, click on the corresponding data set (PreDataSet), you can in the bottom of the pre-processing data set attribute area (Job Properties) to see each PreDataSet contains WorkPlane, Grid, Contact, Detonation, Solution and Output six properties tab (Figure 2.18), activate these property pages can be completed work plane set Meaning, background grid settings, contact control, definition of the starting point (face), solution settings, output options and other operations.

图 2.18: 数据集属性选项卡

2.3 Material definition

PeneBlast has two ways to provide material data to the current task - to create new materials and to load existing materials. Simultaneously, Users can copy or modify the material in the material library (model and parameter values can be modified), and modify the results as New material data is stored in the user-specified library. Each material has a name, density, EOS type, strength mode type, failure model, and parameter source.

1. By creating, loading, modifying, copying, deleting, saving, viewing and other operations, the new definition of this task will be used Material model and the corresponding data, and the name of the new material listed in the material list.

2. PeneBlast will be commonly used material data entry, stored in an existing library (extension mlb), storage

The path is "installation path / bin"; users can choose to use a material directly from the library and import it into the current task, the imported material name will be displayed in the material list.

Use the process description:

To define a material for a current task, use the New, Load, Modify, Copy, delete, save, review, library, and other buttons, perform the appropriate action to complete the material definition work in the task. Click "Material" on the Models menu to pop up material definition dialog (Figure 2.19) to start the new material.

图 2.19: Material Definition 操作界面

The function and operation method of each function key are as follows:

2.3.1 New Material
Function: Create new material and set material parameters.

Operation flow: click on the material definition interface on the New, pop-up "Material Data Input" dialog box (Figure 2.20) Show the name, density, intensity model, state equation, failure model, and parameter source of this material. Different strength modes, type, state equation, failure model has different parameters, the program will be based on the selected model to start a different parameter definition box.

The parameters used for each model can be found in the "MPM3DPP User's Manual". Click on the interface with press button to shrink / expand the corresponding parameter box. When you click Apply, the system will automatically check all the required options are filled out. As long as Material Data Input If the required parameters of the interface are not filled in, click Apply will pop up the error box. Before the required option is not filled in,

In front of the show, waiting for the user to fill in; fill in the required fill in the key to confirm, then become.

Note: The material definition interface, all the parameters of the units have been given a clear, the user input the corresponding variable value must be fine, It is converted to match these units on the interface. In normal calculations, the PeneBlast system automatically converts these values into the user selected unit system to match the value of the variable to calculate
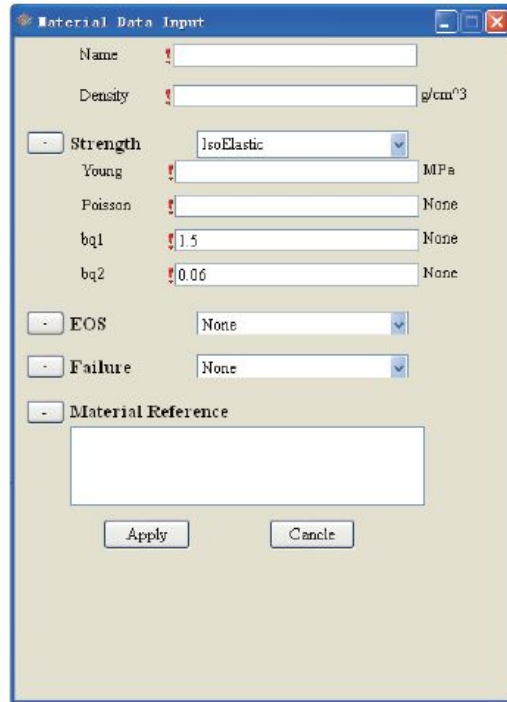
图 2.20: Material Data Input 对话框

2.3.2 Load

Function: Import the existing material model and the corresponding parameters directly from the default (specified) library.

Operation flow: Click the Load button on the material definition interface to pop up the "Load Material Model" window Figure 2.21), which lists the strength of the material in the system library, the state equation, the failure mode Type, the default path for the library file is "X: Install folder / Tsinghua University / PeneBlast / bin". through Over the mouse click to select a material (highlight the material at the same time), click the Load button, you can select the material included n the material list; you can click on the multiple material, click the "Load" button to load this one at a time Materials, and finally click Ok button to confirm.
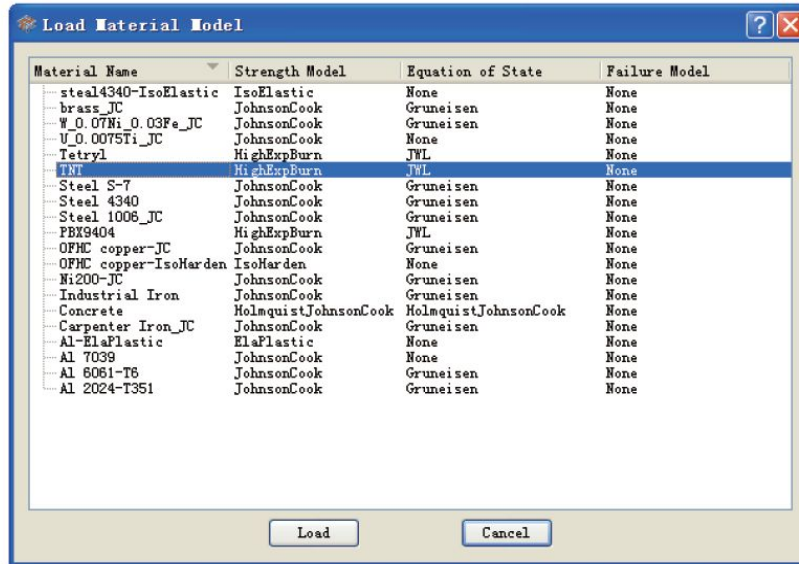
图 2.21: Load Material Model 对话框

2.3.3 Modify

Function: Modify the parameters of the material selected in the material list (shown in Figure 2.22).
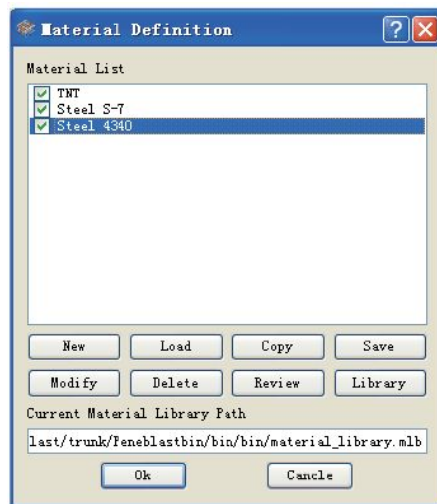
图 2.22: 选定材料列表框中某一材料

Operation flow: first select the material list box in a material (highlighted, as shown in Figure 2.22), then click Modify Pop-up material parameters dialog box (shown in Figure 2.23), modify the variable parameter value, or change the material strength model, the

state enter the new variable value, click Apply to save the changes, close the dialog box.
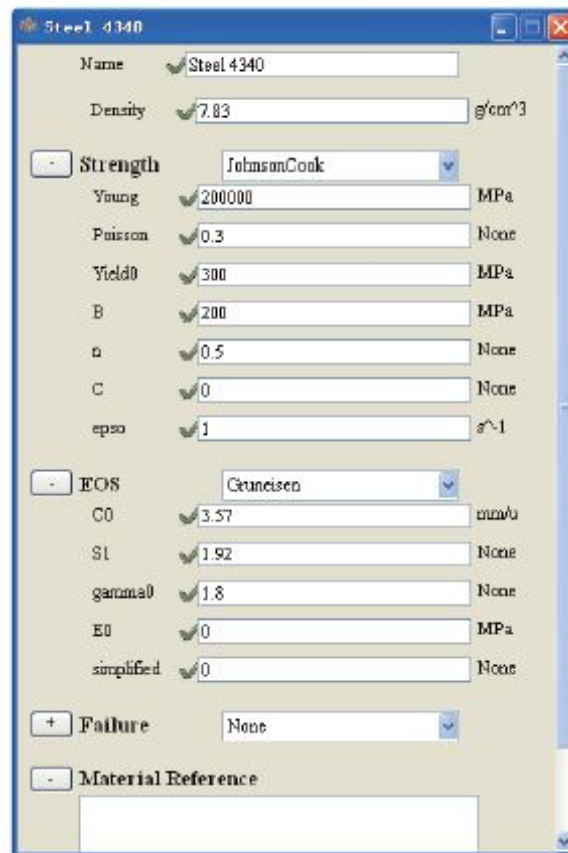
图 2.23: 修改材料数据对话框

2.3.4 Review
Function: View the parameters of the selected material.
Operation flow: first select the material list box in a material (highlighted, as shown in Figure 2.22), then click Pop-up material parameters dialog box (Figure 2.24), then all the material parameters are gray, the user can not modify, click the Cancel off
Close the dialog box.

2.3.5 Copy
Function: Copy the selected material from the list of materials into a new material. The new material is named "raw material name -ewew".
Process: Click Button, in the material list box to create a new material, the new material to copy all the raw materials Type and parameter values, named "raw material name -new", as shown in Figure 2.25.

24

2.3.6 Delete

Function: Remove the selected material from the material list box. Operation flow: first select the material list box in a material (highlighted, as shown in Figure 2.22), then click Button to complete the operation from the material list to remove the material.

2.3.7 Save

Function: Save the selected material in the material list to the specified material library. Process: Click

Button, pop-up Save Materials dialog box, as shown in Figure 2.26; selected (multiple choice)Material in the box (highlighted), and then specify the material into the material library (the current database, the existing

His database and the new database), click Apply to complete the follow-up, click Finished to exit.The default option saves the material information in the current library Current library (Figure 2.19 shows the current material

The current path of the library); if selected into the existing material library Existing library, the bomb File dialog box (Figure 2.27), select the specified path under the library file, click the "open" button to store the material Into the specified library file; if you choose to save the new library New library, the pop-up dialog box, specify the road Diameter, name the name of the new library (the program automatically adds the file extension "mlb"), the operation interface shown in Figure 2.28, the operation into After the pop-up prompt box (Figure 2.29) prompted the material into the success.

2.3.8 Library

Function: Select the library file required for the current task, and the Load operation will be from the library file specified by Library Loading material.

Process: Click Pop-up Select Material Library File dialog box, by specifying the path, select The material library file for the current task, click Ok to select the library, the PeneBlast system sets it as the material and displays its path in the "Current Material Library Path" property box (see Figure 2.30).

2.4 New components

Function: Create a new component under the current dataset.

Process: Click the PreDataSet to activate the Job List Job, and then click the menu bar Models option (see Figure 2.9) In the "Creat Component", you can in the current job pre-processing data set below a new component (default name component1, the new multi-component system automatically numbered in order, the user can double-click the entry to modify the name), such as As shown in Figure 2.31. *Shortcut - the user can right click on a Job's PreDataSet, right click pop-up menu You can create a new*

component, as shown in Figure 2.32. Click Activate the component to repair it in the Component Properties area Change the Component property page.

Component has three property pages in the Job Properties area (Figure 2.33): Properties, Display, and Fill, used to display the properties of the Component (Body list), display mode and fill material, load the initial conditions.

Properties page The Properties list displays all the bodies contained in the Component.

Property page Display and Body Display property page is roughly the same, but this time through the "Bg color" to repair Change the background color of the GUI real-time display area. The property page Fill (Figure 2.34) is used to fill the material of the Component as a whole, All of the Body is filled with a material that loads some sort of initial speed. Select Material, Velocity, or Adaptive split, you can modify all the components within the Body of the corresponding attributes, and finally click Apply, the selected attributes will be applied to All of the components in the Component.

图 2.33: Component Properties 页   图 2.34: Component Fill 页

(translator's note: Fig 2.34 was blank in original manual)

PeneBlast's contact algorithm handles contact between components, rather than between objects. The component name specifies the substance Which component

belongs to. A component consists of multiple bodies. The xmp file uses the component number (starting at 1) to define the group, The PeneBlast system automatically converts the component names into the component numbers in order when the xmp file is output.

2.5 Create a geometric entity

Create a new geometry entity (Block, Cylinder, Ogive, Sphere, User-Defined) in the current component and set Its attributes (space position, geometry, work plane, fill material, object density, initial speed, load setting, display Display format, particle spacing, discrete mode, etc.), see MPM3D User's Manual. For MPM3DPP program The existing body type (Block, Cylinder, Ogive, Sphere), in the xmp file output the body of the space Location, geometry and particle spacing, etc .; for the User-Defined type, PeneBlast system in accordance with the input file The body will be separated into the material into the particle, and then MPM3DPP Point format will be discrete particles after the particles The information is written separately as a file output. All of the newly created objects belong to a selected component in the current preprocessing dataset and are listed in the order in which they were created The following. body created, you can in the Job Browser interface (Figure 2.31), double-click to change the body name, right-click delete Except the body. The interface for creating objects is shown in Figure 2.35.
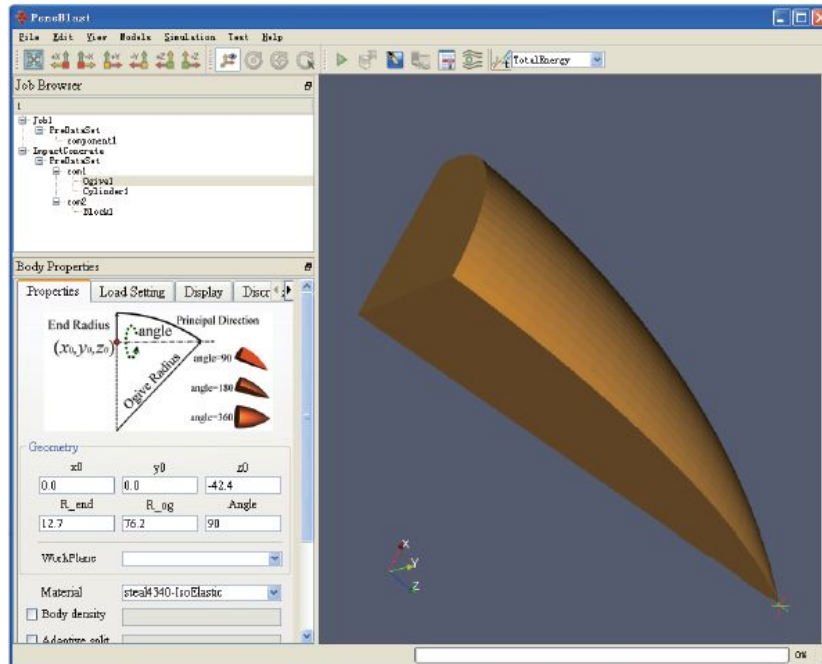
图 2.35: Peneblast 中新建几何实体界面

Operation flow: in the Job Browser in the current PreDataSet selected components below the current Body name (The user can double-click the entry to modify the name), PeneBlast system according to the type of body to show the Body Preview and define the way. For each Body (Block, Sphere, Cylinder, Ogive, etc.) under Component, Job Browser below the Body Properties column contains the body of the four property pages (Properties, Load

Setting, Display, Discretization). The user can define the size of the Body and its properties in the Properties property page It is related to the properties until the completion of the definition of the operation of the geometry.

2.5.1 Block
Function: New cuboid.
Properties page As shown in Figure 2.36
• (x0, y0, z0)
Defines the spatial location of Block. The space coordinate value is relative to the

loaded WorkPlane if Block is not loaded Into any WorkPlane, this set of spatial coordinates that define the location of the Block in the global coordinate system.
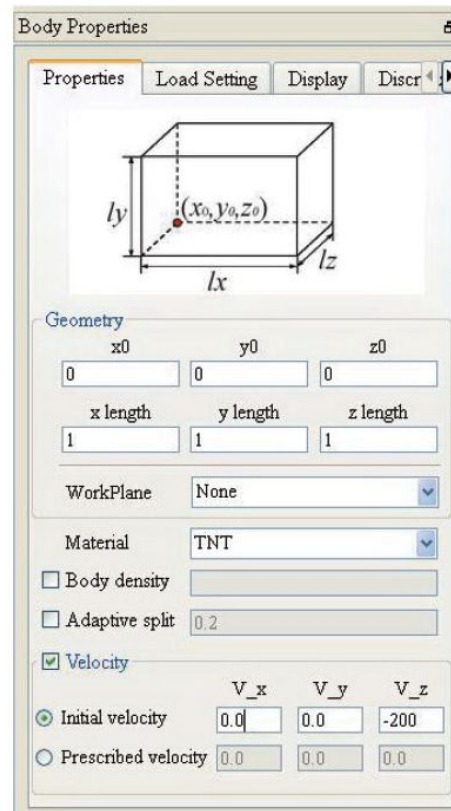
图 2.36: Block 的 Properties 页

• **x length, y length, z length**
Defines the geometry of the Block
• **WorkPlane**
Defines the WorkPlane that Body needs to load. For the definition of WorkPlane, see Section 2.6.1.
• **Material**
Defines Body-filled material
• **Body density**
Define the current density of the body, which is inconsistent with the density of the material used. In a state of compression or expansion.
• **Adaptive Split**
Used to set the adaptive splitting factor
• **Velocity** defines the initial speed of the Body, as determined by the Initial velocity and Prescribed Velocity Fill in value. Initial Velocity is used to set the initial speed of the

current object:

V_x: Sets the speed of the object in the x coordinate direction
V_y: Sets the speed of the object in the y coordinate direction
V_z: Sets the velocity of the object in the z-coordinate direction

Prescribed Velocity is used to set the constant velocity of the current object. At any time, the object is given a constant speed The degree of movement. If the constant velocity value specified for the object is 0, the object corresponds to a fixed rigid body. The element
Has the following attributes:

- V_x: Sets the constant speed of the object in the x coordinate direction
- V_y: Sets the constant velocity of the object in the y coordinate
- V_z: Sets the constant velocity of the object in the z-coordinate direction

Load Setting page
The Load Setting page (Figure 2.37) is used to set the Load properties of the Body
• PrescribedAcceleration Set the time course of the load, the current body with the time to load the load $f\_i = A\_i \times a\_i$. (About MPM3D For a further description of the time history load, see the MPM3D User's Manual.)
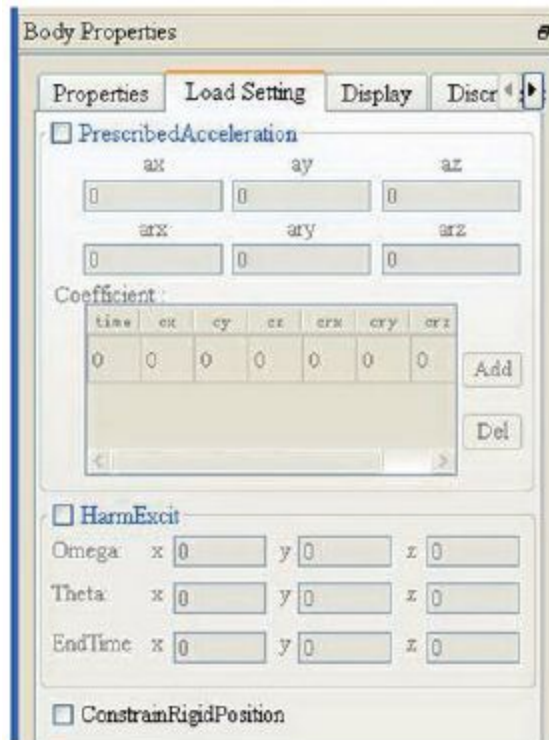
图 2.37: Block 的 Load Setting 页

- ax: Sets the magnitude of the acceleration of the object in the x coordinate direction
- ay: Sets the magnitude of the acceleration of the object in the y coordinate
- az: Sets the acceleration amplitude of the object in the z-coordinate direction
- arx: Sets the angular acceleration of the object in the x coordinate direction
- ary: Sets the angular acceleration of the object in the y coordinate
- arz: Sets the angular acceleration amplitude of the object in the z coordinate direction

Coefficient: used to define a certain time the acceleration multiplied by the coefficient; the element has the following seven attributes:

1. time: set the time;
2. cx: sets the coefficient of ax;
3. cy: set the coefficient of ay;
4. cz: set the coefficient of az;

5. crx: sets the coefficients of arx;
6. cry: Sets the coefficient of ary;
7. crz: set the arz coefficient;

- **HarmExcit (HarmonicExcitation) -** Set the harmonic excitation load, the load function form is as follows:

$$f = A\ sin(\omega t + \theta)$$

And can control the frequency of the load applied in each direction, the phase angle and the action time. The element has three child elements, Each element has three attributes, namely x, y, z. The details are as follows:

- **Omega**: sets value of the harmonic stimulus applied in the three directions of x, y, z;
- **Theta:** sets the value of the harmonic stimulus applied in the three directions of x, y, z;
- **EndTime**: sets the stop time of the harmonic stimulus applied in the three directions of x, y, z;

ConstrainRigidPosition
Set whether the rigid body moves at the specified speed and load.
When the rigid body in accordance with the specified movement law movement, if the collision with other objects are not affected, other objects are subject to
Rigid body effects

Display page
The Display page (Figure 2.38) is used to set the display properties of the Block
- **Visible**
  Whether the object is displayed
- **Zoom to data**
  Button to scale the body in the best size to the GUI real-time display area centered
- **Surface color**
  Defines the display color of the object's surface
- **Edge color**
  Defines the display color of the edges of the object.

- **Point color**

  Defines the display color of the object corner.

- **Representation**

  Defines the rendering of objects (Surface, Wireframe, Surface with edge, Points).

- **Line width**

  Defines the width of the lines in the Wireframe and Surface with edges

- **Point radius**

  Defines the display radius of the corner point.
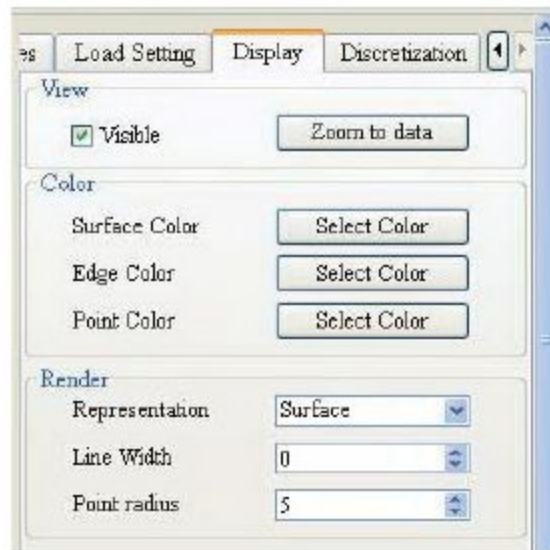
图 2.38: Block 的 Display 页

Discretization page

The Discretization page (Figure 2.42) defines the particle's discrete approach to the block.

- dp

  Block using orthogonal isometric coords, so we only need to specify a direction of the particle spacing dp can be.
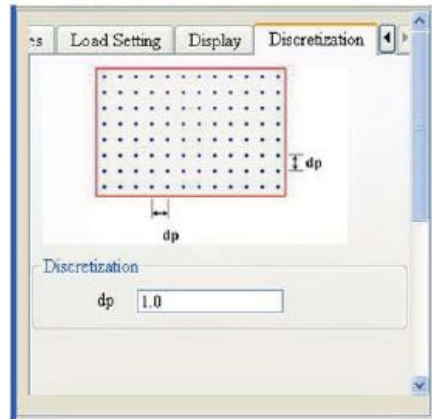
图 2.39: Block 的 Discretization 页

2.5.2 Cylinder

Function: Newly used regular rotating body, including the cylinder, cone, round table, the model of the schematic diagram shown in Figure 2.40 illustrates.
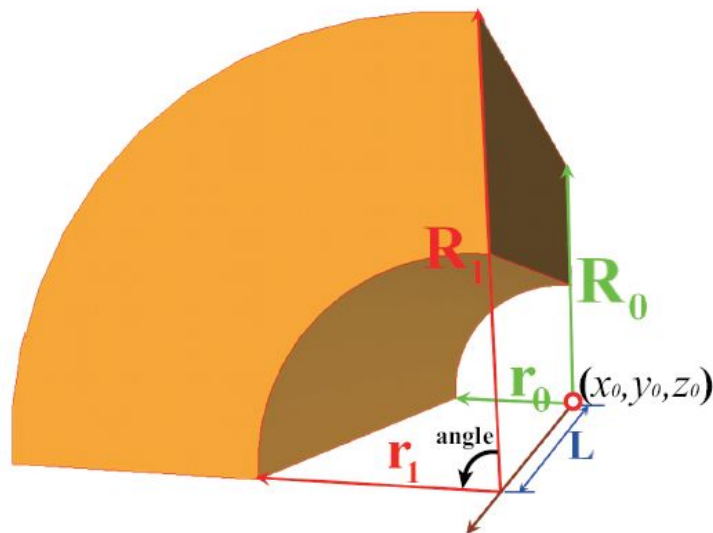
图 2.40: Cylinder 模型参数示意图
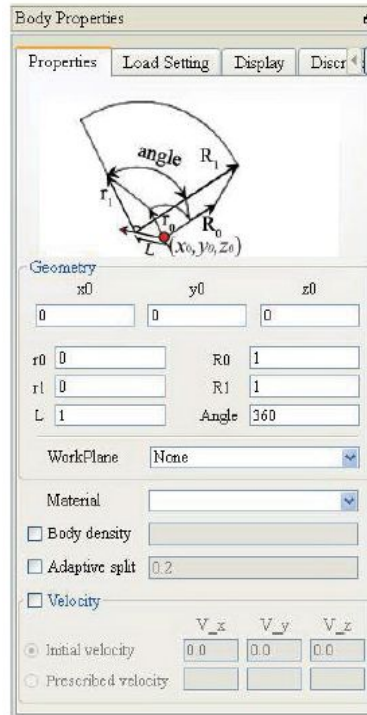
图 2.41: Cylinder 的 Properties 页

Properties page

As shown in Figure 2.48

- **x0, y0, z0** - Defines the spatial position of the Cylinder starting point. The space coordinate value relative to the loaded WorkPlane, ifCylinder does not load any WorkPlane, this set of spatial coordinates that defines Cylinder in the global coordinate systems position.
- **r0** - The cylinder inner diameter of the starting face
- **R0** - The cylinder outer diameter of the starting end face
- **r1** - Cylinder ends the inner diameter of the end face
- **R1** - Cylinder ends the outer diameter of the end face
- **L** - The axial length of Cylinder
- **Angle** - Defines the angle of rotation
- **Workplane, Material, Body density, Adaptive split, Velocity** -The meaning of the these edit box parameters is the same as in aforementioned Section 2.5.1 properties, not repeat here.

Load Setting page

The meaning of the edit box parameters in the Crowinder's Load Setting tab is the same as described in Section 2.5.1, Load Setting page.

Display page

The meaning of the edit box parameters in the Cylinder's Display tab is the same as in Section 2.5.1 Display page.

Discretizaion page

The Discretizaion page (Figure 2.42) defines the Cylinder's Discrete approach.

- **dp bottom** - specifies the particle spacing of the starting end of the Cylinder
- **dp top -** specifies the particle spacing of the end face of Cylinder.
  dp bottom, dp top After the determination of the layers along the height direction (Z axis positive direction) of the particles, the spacing by the ratio
  Relationship changes.
- **dp Discretization** - discrete mode, the value of 0 or 1: 0, orthogonal way of distribution; 1, the way to the distribution point.
  Due to the geometrical characteristics of the rotating body itself, the use of a circumferential distribution (as shown in Figure 2.43) can better keep the rotating body geometric shape.

  - If the Angle property of Cylinder on the Properties page is not 90, or 180, or 270, or 360, PeneBlast The system only supports loopback
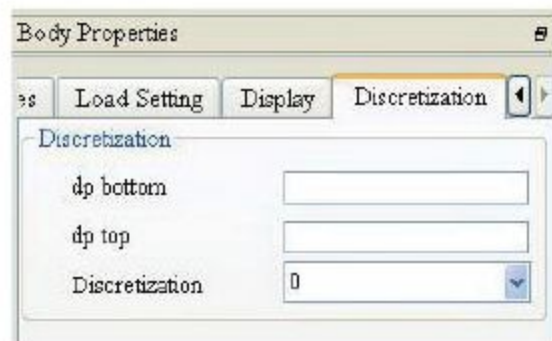
图 2.42: Cylinder 的 Discretization 页

2.5.3 Ogive

Function: New oval body, the default warhead direction along the Z axis positive

direction.

Properties page As shown in Figure 2.44

- **x0, y0, z0** - Defines the spatial location of the Ogive starting point. The space coordinate value relative to the loaded WorkPlane, if Cylinder Does not load any WorkPlane, this set of spatial coordinates that define the Cylinder in the global coordinate system in the bit set.
- **R_end** - The end radius of the oval body.
- **R_og** - Ogive Radius of oval body.
- **Angle** - Define the opening angle of the new oval
- **Workplane** - Material, Body density, Adaptive split, Velocity

    The meaning of the above edit box parameters with the aforementioned Section 2.5.1 Properties page, not repeat here.

图 2.43: Cylinder 环向布点示意图

图 2.44: Ogive 的 Properties 页

Load Setting page

The meaning of the edit box parameters in the Ogive Load Setting tab is the same as described in Section 2.5.1, Load Setting

page.

Display page

The meaning of each edit box parameter in Ogive's Display tab is as described in Section 2.5.1 Display page.

Discretizaion page

The meaning of the edit box parameters in the Discretizaion tab of Ogive is the same as that described in Section 2.5.2 Discretizaion page.

2.5.4 Sphere

Function: New sphere. Properties page As shown in Figure 2.45.

- **x0, y0, z0**

  Defines the spatial position of the sphere. The space coordinate value relative to the loaded WorkPlane, if Cylinder does not load any WorkPlane, which defines the position of Cylinder in the global coordinate system.

- **Radius outer**

  Ball outside diameter.

- **Radius inner**

  Ball inside diameter.

- **Angle** -The angle of the two semicircular surfaces of the ball - type rotating body.

- **Half** - This property is only available when Angle = 90. When Angle = 90 check the property that whether to take the

  1/4 of the upper part of the ball: that is, half = 0, to generate a quarter ball; when half = 1, take the first half of the quarter points, that is generated within the first divination limit of 1/8 ball.

- **Workplane, Material, Body density, Adaptive split, Velocity**

  The meaning of the above edit box parameters with the aforementioned Section 2.5.1 Properties page, not repeat here.
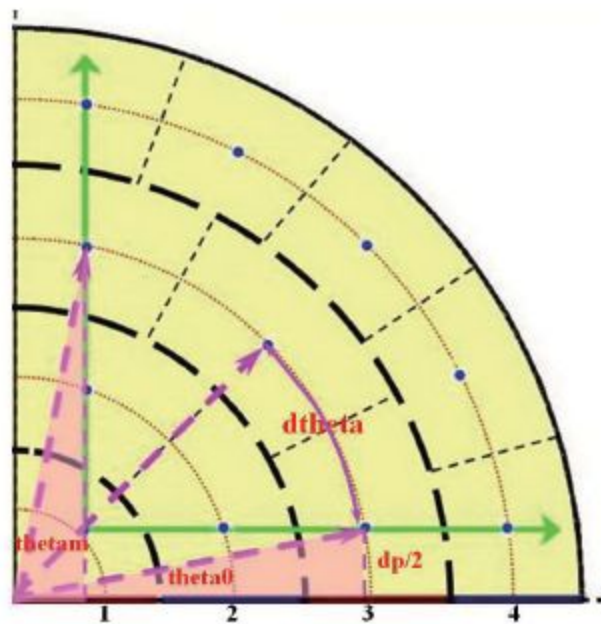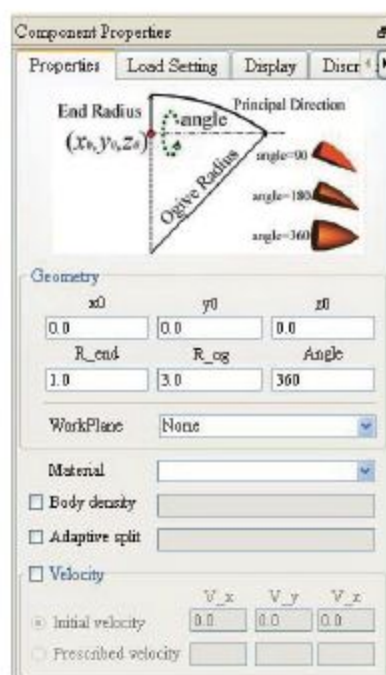
Load Setting page

The meaning of the edit box parameters in the Load Setting tab of Sphere is as described in Section 2.5.1, Load Setting page.

Display page

The meaning of the edit box parameters in the Display tab of Sphere is as described in Section 2.5.1 Display page.

图 2.45: Sphere 的 Properties 页

Discretization page

The Discretization page (see Figure 2.46) defines the particle's discrete approach. Due to the geometric properties of the sphere itself. The ball can be discretized in different ways, such as the orthogonal distribution (Discretization = 0) and the directional (Discretization = 1). Orthogonal distribution in three mutually perpendicular directions (ie, along the x, y and z axes) pairs.The sphere is orthogonal to the dispersion, and the way of the distribution of the ball along the spherical coordinate system of the three coordinate axes on the direction of the ball to the equidistant distance Scattered, this way can be better to keep the spherical surface.

● **dp -** Sphere no matter what kind of distribution method, only need to specify a reference particle spacing dp can be.
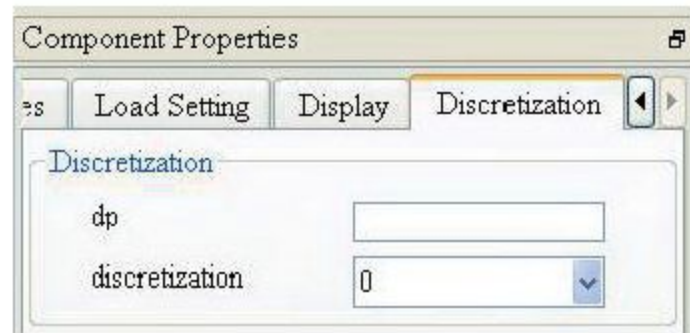
图 2.46: Sphere 的 Discretization 页

2.5.5 Polyhedron

Function: Create a new entity of the 3D geometry model represented by the .stl file. Properties page As shown in Figure 2.47.

● **x0, y0, z0** - Defines the spatial position of the polyhedron. The space coordinate value is relative to the loaded WorkPlane if the polyhedron is not loaded
Into any WorkPlane, this set of spatial coordinates that define the polygon in the global coordinate system position.
● **Scale** - Defines the scale of the imported model
● **Workplane, Material, Body density, Adaptive split, Velocity**
The meaning of the above edit box parameters with the aforementioned Section 2.5.1 Properties page, not repeat here.

Load Setting page

The meaning of the edit box parameters in the Polyhedron's Load Setting tab is the same as described in Section 2.5.1, Load Setting page.

Display page

The meaning of each edit box parameter in Polyhedron's Display tab is as described in Section 2.5.1 Display page.
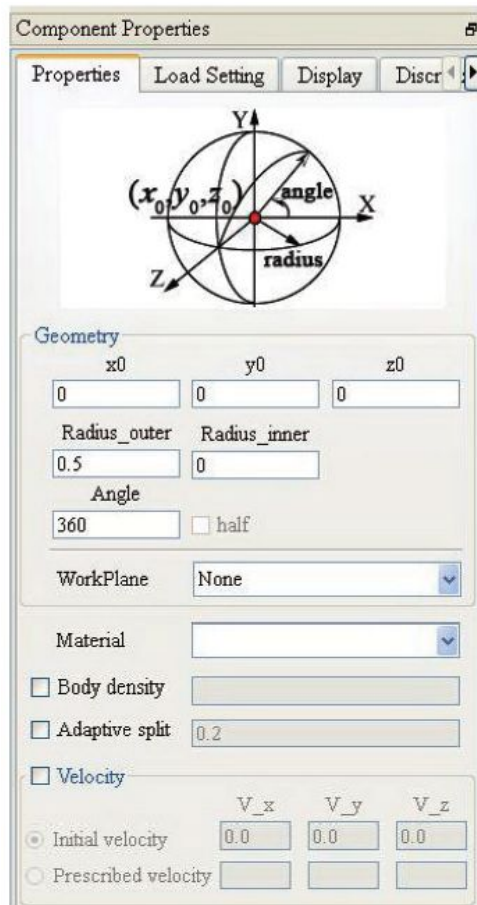
Discretization page

The Discretization page (Figure 2.48) defines the polygon discrete approach. As the ball is introduced into the three-dimensional geometry body is often more complex, can not

use a unified formula for discrete point coordinates calculation, so here the use of geometric uniform inside and the external use of ball filling algorithm based on the layout of the program, this approach can ensure that the internal discrete uniform, but also can be better to maintain the polyhedral surface.

- **dp global**
  Defines the reference value of the global discrete point spacing, and the actual spacing may fluctuate up and down at this distance
- **Test Solidfill**
  Click this button to save the discrete effect before saving the job (if the model geometry is changed, you need to click this again Button to discrete, save the job does not automatically discrete operation), discrete data exists button left edit box
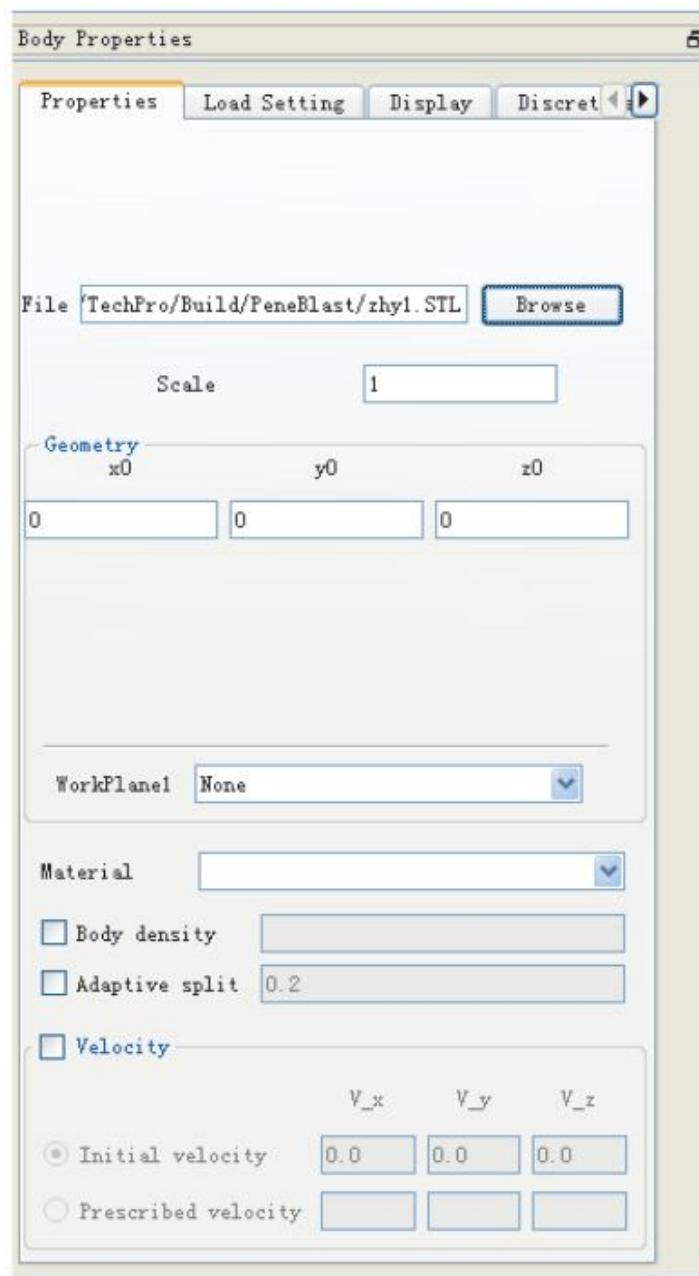
Polyhedron Properties

图 2.47: Sphere 的 Properties 页

2.5.6 User-Defined

Through the third-party CAE modeling software, to construct complex geometric entities, and discrete into a finite element mesh form (head before only supporting the eight-node hexahedron unit), the Grid2Point tool of the PeneBlast system is implemented by importing software from CAE. A file containing the entity node and cell

information generated to generate a text file containing material point information. PeneBlast. The system creates new complex shape geometry by importing the text file containing the substance point information.
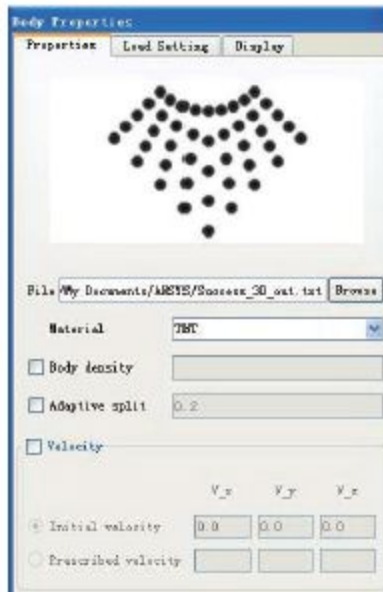
图 2.49: User Defined Entity 的 Properties 页

Properties page

As shown in Figure 2.49

- **File**

  Specifies the path of the text file that contains the substance point information.

  The text file containing the substance point information lists all the mass information contained in the entity,

  ○ the first column: the material quality.

  ○ The second to fourth columns are the spatial coordinates x, y, z of the material point.

  The format is as follows:

  1.0079665e + 000 0.0000000e + 000 0.0000000e + 000 0.0000000e + 000

  9.3025750e-001 3.4641000e + 001 2.0000000e + 001 0.0000000e + 000

  1.8897898e + 000 1.7321000e + 000 1.0000000e + 000 0.0000000e + 000

44

...
    5.2774400e + 000 1.5385000e + 001 1.7252000e + 001 0.0000000e + 000
- **Material, Body density, Adaptive split, Velocity**
  The meaning of the above edit box parameters with the aforementioned Section 2.5.1 Properties page, not repeat here.

Load Setting page

The meaning of the edit box parameters in the Load Setting tab of User-Defined is as described in Section 2.5.1 Load Setting page.

Display page

User-Defined Display tab of the edit box parameters in accordance with the above Section 2.5.1 Display page description fill write. After the parameters are filled in, click Apply, the GUI real-time display will show the imported entity, as shown in Figure 2.50 When the system creates a file containing the geometric information for all the material points of the disc under the Job storage path of the task.
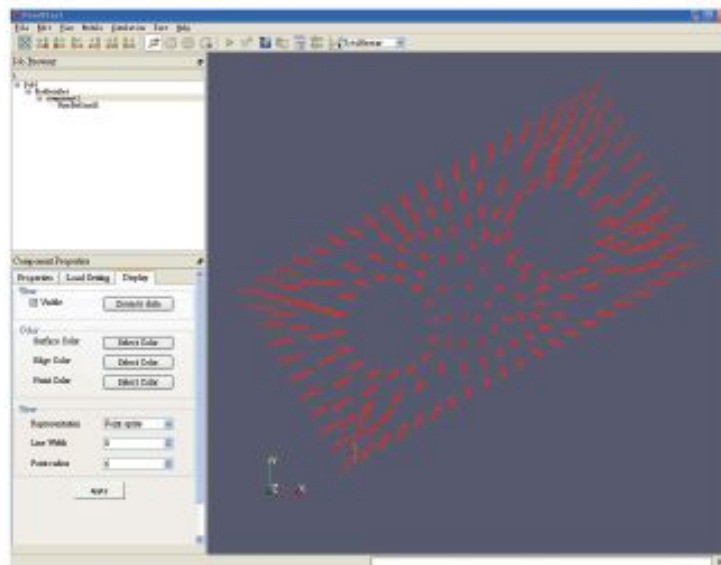

Figure 2.50: GUI Graphical Area Real-time
display of user-defined geometry entities

2.6 Set the properties of the pre-processed data set

The Job Properties of the Preprocessing Data Set has the property pages WorkPlane, Grid, Contact, Detonation, Solution and Output, used to set the local work plane parameters, background grid information, contact control, the definition of the starting point (surface), output options, solution settings. In the Job Browser, click on a preprocessing dataset in the list to preprocess the dataset. The PreDataSet Properties will display the property pages **WorkPlane, Grid, Contact, Detonation, Solution** and **Output.**

2.6.1 WorkPlane property page

WorkPlane property page (shown in Figure 2.51) for the user to enter the specified parameters used to define a set of local coordinate system, The user establishes the Body model in these local coordinate systems. In MPM, the coordinates of the particle points are in the global coordinate system definition, and for the body loaded WorkPlane, the initial position parameters (x0, y0, z0) of the reference system Is a local work plane. In order to achieve a variety of Body rotation and translation operations, the need to introduce the work plane will be the body. The discrete particles are moved from the local system to the global coordinate system. In the local coordinate system to establish the body discrete,

The use of local work plane can be used to complete a single operation of the various body rotation and translation, the Body moved to the air between the specified location. Load the WorkPlane on the Body to get the location of its particle points in the global coordinate system.
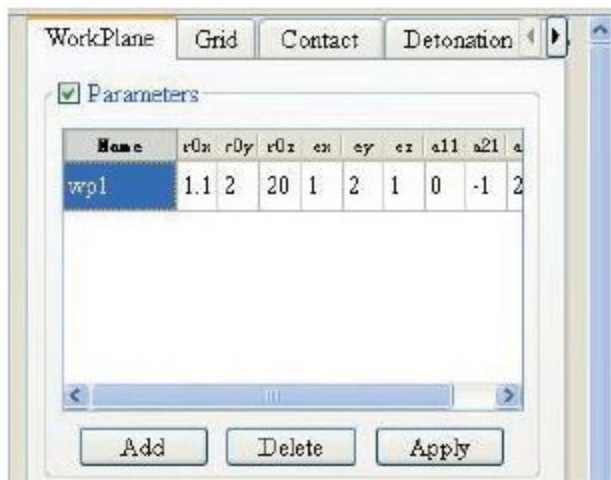

图 2.51: WorkPlane 属性页

- "**Add**" button
  Create a new set of work planes.
- "**Delete**" button
  Deletes the currently active set of work planes.
- "**Apply**" button
  Save all the parameters in the current WorkPlane list.

Each work plane contains the following four sets of information:
- **Name**
  WorkPlane name, each new new work plane, the default name will be automatically numbered, the user can double-click the default name of the edit box, modify the new name.
- **r0x, r0y, r0z**
  The offset of the origin of the WorkPlane relative to the origin of the global coordinate system
- **ex, ey, ez**
  The Z-axis vector of the local coordinate system (WorkPlane) is in the direction vector of the global system
- **a11, a21, a31**
  The X-axis vector of the local coordinate system (WorkPlane) is in the direction vector of the global system.

Precautions:
(1) vectors (ex, ey, ez) and (a11, a21, a31) are used to specify the Z-axis (main direction) and X-axis of the local coordinate system, respectively. The amount of square under the global system, do not require a unit vector (that is, not required)

$$\sqrt{e_x^2 + e_y^2 + e_z^2} = 1 \quad \text{and} \quad \sqrt{a_{11}^2 + a_{12}^2 + a_{13}^2} = 1 \text{ (optional)}$$

but what is required is that the two directions are vector orthogonal, i.e., that

$$e_x a_{11} + e_y a_{12} + e_z a_{13} = 0 \quad \text{(required)}$$

must be strictly satisfied. If the body Loaded this illegal work plane, will pop up the error box as shown in Figure 2.52.
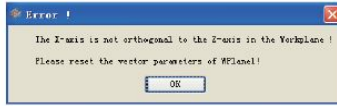
图 2.52: WorkPlane 错误提示框

(2) The Y-axis direction of the local coordinate system is automatically obtained by the vector (ex, ey, ez) by the vector (a11, a21, a31), such that the WorkPlane's coordinate frame is still maintained for the right hand system.

2.6.2 Grid property page

The Grid property page (Figure 2.53) is used to set the background grid parameters, including the basic grid parameters, moving the grid and moving state grid settings.

- **x0, x1, y0, y1, z0, z1**
  Define the background grid area for solving the problem.
- **Boundary Type**
  Set the boundary conditions of the boundary plane, can be set to (Free / Fixed / Symmetric / Transmit / Rigid Plane), points Do not match (0/1/2/3/4):

- **boundary = "0"** - free boundary interface;
- **boundary = "1"** - fixed boundary, constrains all the degrees of freedom of the boundary node;
- **boundary = "2"** - symmetric boundary, constraining the normal motion of the boundary node, the other two directions free;
- **boundary = "3"** - transmission boundary surface, that is, without reflection boundary conditions;
- **boundary = "4"** - rigid surface, the boundary node can not penetrate the boundary in the normal direction, the other direction free.
- **dcx, dcy, dcz**
  x, y, z in the three directions of the initial grid spacing
- **Mass cutoff**
  Set the minimum node quality cutoff coefficient c, the default value for the system can represent the minimum error DBL_EPSILON Or FLT_EPSILON. (See "MPM3DPP User's Manual");
- **GIMP**
  Whether to use GIMP algorithm

48

- **Consider Loss Particle**

  Set whether or not the dead particles are included in the area where the mass point is counted. If checked, it means that the invalid particles are counted and unchecked

  That does not count into the failure of particles. The default is not counted as invalid particles.

- **Time step factor**

  Set the mobile grid interval step factor factor. Taking into account each step to determine whether to move the grid to spend more time,

  So move the grid by calculating how many steps are predicted. The program multiplies the calculated number of steps by factor

  The value obtained by dt scale is the number of interval steps that are actually used in the program. factor The default value is 0.8.

- **XMin**

  Sets the x-direction minimum moving boundary of the moving grid

- **XMax**

  Sets the x-direction maximum moving boundary of the moving grid

- **YMin**

  Sets the y-direction minimum moving boundary of the moving grid

- **YMax**

  Set the y to the maximum moving boundary of the moving grid

- **ZMin**

  Sets the z-direction minimum moving boundary of the moving grid

- **ZMax**

  Sets the z-direction maximum moving boundary of the moving grid

- **Clear cell step**

  Sets the number of steps to clear the original grid and node instances. Every time a certain period of time to clear the original grid and An instance of a node that is rebuilt when used to save memory. The default time interval is 10, if the value is set

  Is less than or equal to 0, it means never clear. Valid only if Dynamic cell is checked.

2.6.3 Contact property page

- **Contact on**
Whether to enable contact, only check the Contact on when the following options are available (Figure 2.54), if the number of components is 1 The touch algorithm is not enabled.
- **Failure to component**
Whether to enable particles as a standalone component after failing, the default is not enabled. After the activation, the failed particles as
The relationship between individual components and other components is calculated using the contact algorithm.
- **Exclude components**
Sets whether or not to exclude components that have a defective trace when calculating the contact surface normal. Check that when calculating the contact normal Regardless of the presence of the defective particle assembly, unless both components in contact have failed particles. Turn off this feature by default.
- **AutoCompute component weight**
Whether to automatically calculate the components of the contact surface normal calculation of the weight. When checked, it automatically sets the normal direction of each component's contact surface Direction of the weight, not checked that does not automatically set, the default is not checked. The method determines the weight according to the elastic modulus In Mooney Rivlin material, the weight is the largest, for other materials without elastic modulus is set to -32767, For failing components (component 0) is automatically set to -32768.
- **Distance type**
Sets the method of calculating the minimum contact spacing factor.
    - **distance_type = "0"** means no calculation,
    - **distance_type ="1"** means to calculate the nearest particle spacing of two objects.
    - **distance_type = "2"** means to calculate the distance between two objects distance. distance_type = "2" if the volume of the particle is taken into account, since the distance_type = "1" Calculate slightly larger.
    - **Min distance** set the minimum contact spacing coefficient dist. In the standard contact method of the material point, when two objects are on the same background. When the quality and momentum of the mesh nodes contribute, the two objects are considered to be in contact, but the distance between the two objects Is not zero, but less than twice the size of the grid cell. In order to overcome this defect, you can set the minimum connection

Touch the distance coefficient dist to determine whether contact has occurred. When two objects are on the same background grid node the quality and movement the amount of contribution and the distance between them is less than dist multiplied by the grid cell size, only that the two objects have occurred contact, Otherwise there is no contact. The default value of min_distance is 0.0, and no contact distance judgment is made.

- **Contact time step**

Whether to correct the time step of the explicit integration according to the contact algorithm. Check the explicit integration time according to the contact algorithm

Step length correction. The default value is not corrected. The contact algorithm will significantly affect the critical time step of explicit integration enable this option.
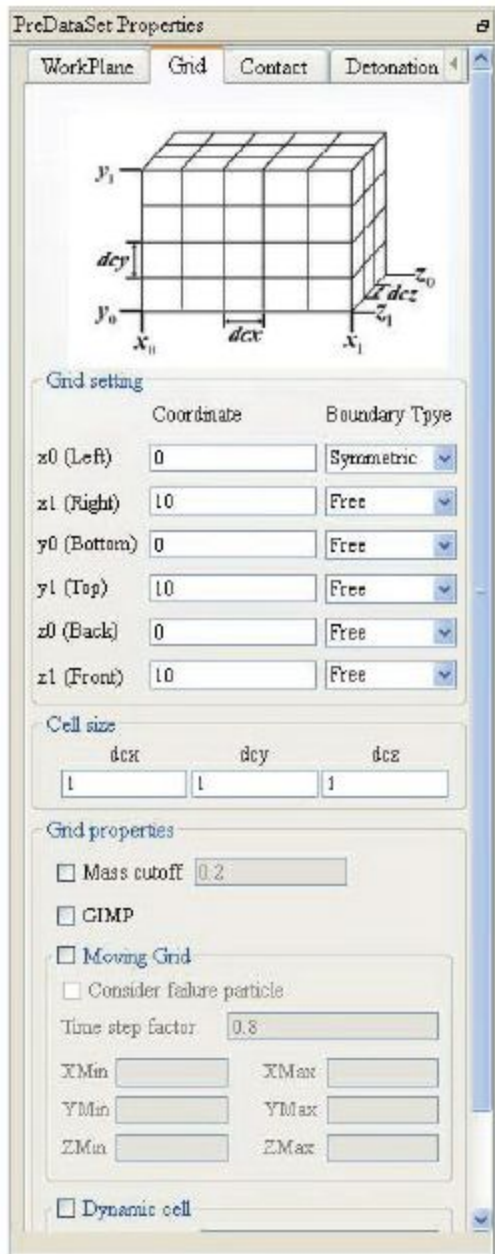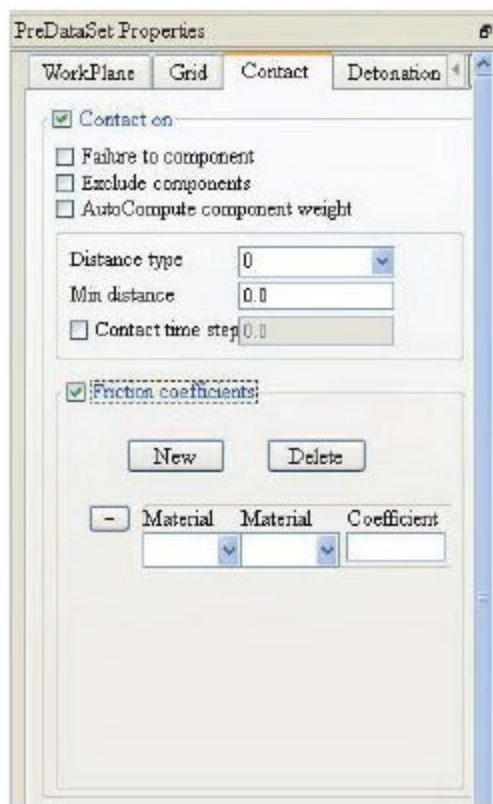
图 2.53: Grid Definition

图 2.54: Contact Option

- **Friction coefficients** - Define the coefficient of friction between materials

"New" button: Defines the friction coefficient between a group of two materials. Two "Material" box were selected under two Material, "Coefficient" Enter the friction coefficient between these two materials.

"Delete" button: delete the current activation of this group of friction coefficient.

2.6.4 Detonation property page

Define the initiation point / surface of the explosive and set its ignition time (Figure 2.55). Click "+" and "-" to expand and shrink the burst point / surface data, while the item is highlighted.

- **"Add" button**
  Create a new point / face
- **"Delete" button**
  Deletes a currently active point / face
- **"Apply" button**
  Save all the parameters in the current Detonation Points / Plans List
- **Type**
  Set the type of burst / face. type = "point" means the definition of the starting point, type = "plane" that the definition of the starting burst face.
- **Detonation Time**
  Define the ignition time of the initiation point / face
- **Detonation point**
  If type = "point", you need to set the following properties:
  - **x, y, z:** used to set the starting point in the global coordinate system position.
- **Detonation plane**
  If type = "plane", you need to set the following attributes:
  - **Direction**: Sets the normal of the starting surface, which is 0, 1, 2, respectively, corresponding to x, y and z axes;
  - **Location**: Sets the position of the detonating surface on the coordinate axis parallel to its normal;
  - **Left:** Sets the left side coordinate value of the detonating surface;
  - **Right:** Sets the coordinates of the right side of the detonation surface;
  - **Bottom:** Sets the bottom coordinate value of the detonation surface;
  - **Top:** Sets the upper coordinate value of the detonation surface.

See Figures below:
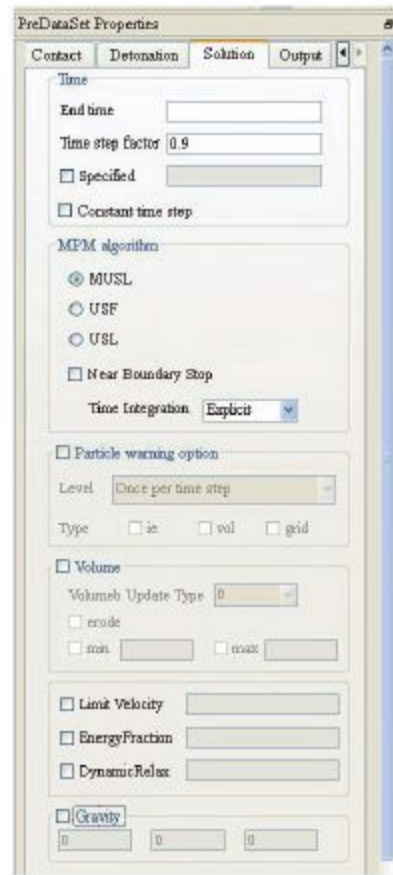
图 2.55: Detonation Option

图 2.56: Solution Option

2.6.5 Solution Properties page

The Solution property page (see Figure 2.56) focuses on the relevant settings for the solution, such as physical simulation time, time step size factor, volume update mode and so on.

- **EndTime**
  Set the physical simulation time
- **Time step factor**
  Set the time step factor, the default value is 0.9.
- **Specified**
  Specify a specific time step and use this as a fixed time step when the program is calculated, making it easier to compare Use, the default is not used.
- **Constant time step**
  Whether a fixed time step is used

- **MPM algorithm**

  This check box is used to set the options for the solution format used in the calculation process:
  - ○ **MUSL, USF, USL:** These three radio boxes are used to select the algorithm format. By default, the "MUSL" format is used.
  - ○ **Near Boundary Stop:** Sets whether to save the restart file and exit when the particle approaches the boundary. Use this function. Mainly in the need to use the state file to restart the calculation of the case considered to use.
  - ○ **Time Integration:** Specifies the integration scheme used by the program. Select "Explicit" that uses explicit points, Select "Implicit" to use implicit integration. The default is explicit. The implicit integral solver is not complete,

    Temporarily unavailable.
- **Particle warning option**: In the process of calculation, due to individual particles appear non-physical phenomena such as negative energy, negative volume, the program should be the end Only calculation. But according to the previous calculation experience, the individual particles of the non-physical scene will not affect the overall accuracy of the calculation, Therefore, the use of such a calculation of the first output warning message, and according to the specific circumstances to take appropriate measures.
- **Level:** Set the warning level. The options are: "Once per timestep", "All warning messages", "Stop when warning, "the corresponding value range is 0,1,2, the default value is 0. Once per timestep (level = "0"): A warning message is output at each time step; All warning messages (level = "1"): A warning is issued when the particle variable satisfies the warning threshold; Stop when warning (level = "2"): Once the particle variable satisfies the warning threshold, the error is reported and the program is terminated.
- **Type:** set the physical quantity of the monitoring, type can be "ie", "vol" or "grid", respectively, control the output negative energy Warning messages, negative volume warning messages, and mass flyout grid area warning messages.
- **Volume:**
  - ○ **Volume Update Type**: Set the way to update the volume of the particle, 0 4, the default value is "0".
  - ○ **erode:** set the erosion volume criterion, if the particle size is less than erode when it is eroded (ie deleted);
  - ○ **min:** set the minimum particle size correction factor, the correction is to limit the particle size in a certain range, the default Do not fix

- ○ **max:** Sets the maximum volume correction factor for the particle. The corrective mode is to limit the particle size to a certain range.
  Not for correction.
- ○ **Limit Velocity:** Set the maximum speed limit for the particle.
  In the calculation process, individual particles may be abnormal, the speed is too large, and thus greatly reduce the time step. You can check this option to limit the maximum speed of the particle.
- ○ **Energy Fraction:** Set the energy score, when the total energy error exceeds this score, the program will error and termination of the calculation. Default is not Start this control option. In the process of calculation, if there is no mass fly out of the grid case, the total energy should be guaranteed to be constant. But During the calculation, the total energy will change due to numerical dissipation or calculation of errors. If the error is due to the total, The amount of energy changes with the total energy ratio of more than a certain score, the program calculation results are not credible, then you can terminate the calculation.
- ○ **Dynamic Relax**: Set the viscous damping coefficient
  The damping is achieved by calculating the external force of the node when it is proportional to the magnitude of each particle point,
  Of course, you can also use the speed of the node to introduce. The current setting is uniformly applied on all bodies.
- ○ **Gravity:** Set the weight of the acceleration in three directions of the component, the default does not open. The unit of gravity acceleration depends on the current position The selected unit system, so the actual value of the user needs to manually convert.

2.6.6 Output property page

The Output property page (Figure 2.57) is used to set options for outputting the results, such as the console monitoring output time Separated, post-processing file type, post-processing observation of physical quantities.
- ● **Console output** Set the relevant options for the console
  - ○ **Time interval:** Sets the console to calculate the status interval. The console monitoring content defaults to the current time, the current time step, the time step, the total kinetic energy, and the total internal energy.
  - ○ **Momentum print:** Sets whether to report momentum on the console.

- **Statistics**

  Statistical memory and actual grid usage. memory: whether to output memory at a specified time interval to .sta text This feature is currently only available under Windows. Check for output, do not check not output. The default is not output.
  - ○ **cell_occupy:** whether to output a grid of particles at a specified time interval to the .sta file. Check this option The output is not checked. The default is not output.
- **Time history output:** Record the variable time-history curve of the observation point, and the variables that can be selected for output are shown in Table 2.1.
  - ○ **"Add" button:** add a watch point, the x, y, z said the observation point in the global coordinate system in three directions Coordinate.
  - ○ **"Del" button:** Deletes the currently active watchpoint.

- **Output variables**

  Set the physical quantities to be observed in the post-processing file. The variables that can be selected for output are shown in Table 2.1.
- **Output format**
  - ○ **Output time interval**: Defines the time interval for post-processing files.
  - ○ **H5Part:** whether to output H5part format post-processing data files;
  - ○ **VTK_Data**: whether to output VTK format post-processing data files.

Table 2.1: Physical properties for each output variable

|					
seqv	Mises stress	velocity_0	x dir. velocity	exx	x dir cumul. strain
epef	equivalent plastic strain	velocity_1	y dir. vel.	eyy	y dir cumul. strain
pres	pressure	velocity_2	z dir. vel.	ezz	z dir cumul. strain
density	density	sigy	yield stress	posx	x displacement
tria	stress triaxiality σ^*	damg	degree of damage	posy	y displacement
fail	failure	engi	deformation energy	posz	z displacement
engk	kinetic energy	body	body number	com	component number
cp	sound speed	kelv	Kelvin temp	mat	material group
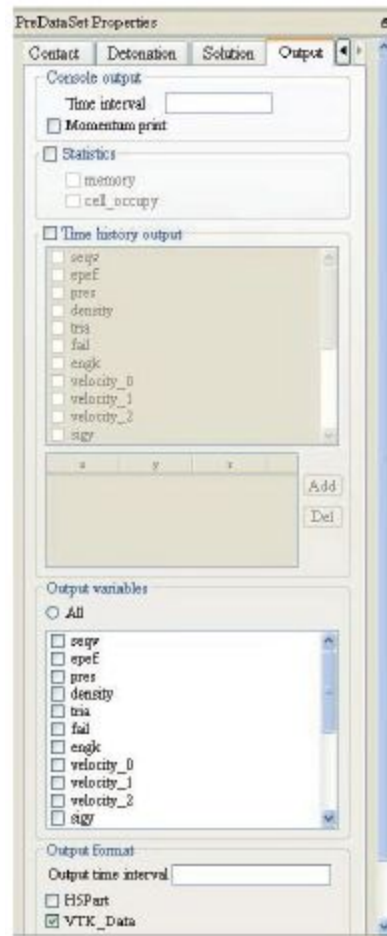
图 2.57: Output Option

Chapter 3 Analysis and Calculation

3.1 Create an example file

PeneBlast system can be generated through the following two ways to directly calculate the study file:

(A) through the pre-processing interface in the operation, set the corresponding parameters to generate In accordance with the previous chapter of the interface function description, in the PeneBlast system pre-processing interface in the new definition of the task, the new material and geometric model, set the solution parameters, click the menu bar File-> Save Job, you can generate the current working path xmp file. The user can also right-click the specified task in the Job Browser and click "Save" to complete the operation. For details, refer to Chapter 5, "PeneBlast Getting Started Guide".

(B) directly into the existing xmp file.
Click the menu bar File-> Load Job, the user can pop up the file dialog box (Figure 3.1), will specify the path of an xmp data file into the PeneBlast system will be loaded into the corresponding parameters, while the entity model Is displayed in the GUI real-time display area. The xmp file holds all the information entered in the pre-processing interface and can be pre-processed system generated, or by the user directly in the text editor to manually generate the generated. Users can import xmp data files. Perform a model check in the pre-processing interface, or reset the parameters.
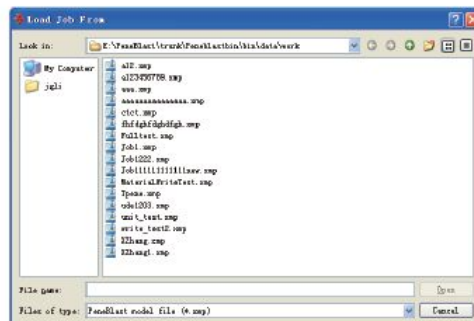

图 3.1: Load Job 对话框

3.2 Perform the calculation

1. Before starting the calculation, click on the menu bar Simulation-> Check Input Data, or click the toolbar On the button to check whether the current saved xmp file conforms to the format requirements and gives the result of the check Error pop-up dialog box (Figure 3.2), given the specific error message; format is correct pop-up dialog box (Figure 3.3) prompt format check.
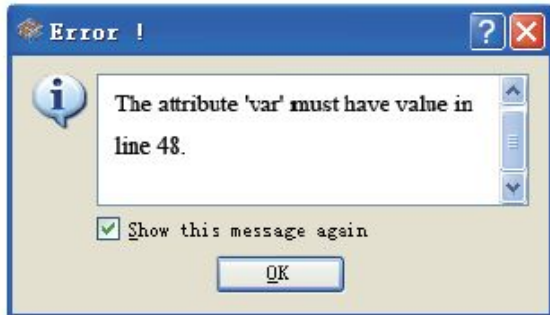

图 3.2: Error Dialog Box

图 3.3: Success Dialog box

2. Click the menu bar Simulation-> Start, you can also click the button on the toolbar, PeneBlast began to enter Line of the current task of the calculation (Figure 3.4).
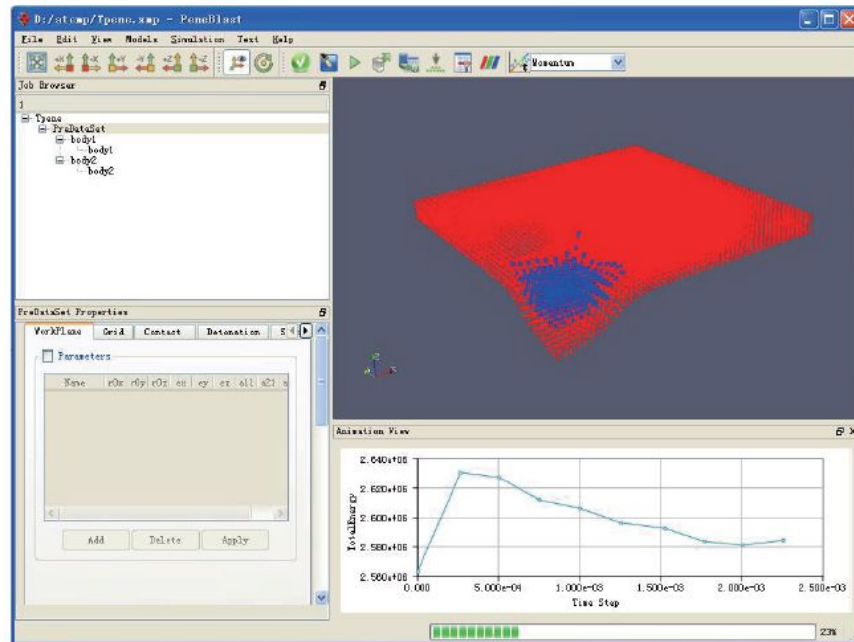
图 3.4: PeneBlast 界面实时显示计算进程

3. During the calculation, the user can pause at any time as needed (Simulation-> Pause, or click on the toolbar button) and terminate the calculation process (Simulation-> Stop, or click the button on the toolbar) to change the calculation real-time monitoring area
The display variable.

4. After the calculation process is paused, the user can click on the menu bar Simulation-> Save RestartFile (or click the toolbar on the button), save the restart state file, the program pops up as shown in Figure 3.5 dialog box, prompts to restart the file save success.

5. To continue the calculation from the interrupt state, click the menu bar Simulation-> Restart, you can also click with the buttons on the bar, the PeneBlast system reads the last restarted state file (* .pe) state to continue to calculate (restart).

6. Click the menu bar Simulation-> Postprocessing, you can also click the button on the toolbar, PeneBlast The system starts calling Paraview for post-processing, and the user will observe the results in the open Paraview interface.

Chapter 4 Postprocessing System

PeneBlast system selection of open source software ParaView as a post-processing software. Since ParaView can run either on a single-processor workstation, you can also run a large computer in a distributed memory, so the ParaView can run either
In a single-processing application, you can also process large data by distributing the data across multiple processors. For ParaView Software, PeneBlast system can generate two types of post-processing data files, the suffixes are vtu and H5Part. Here we offer a brief introduction to some of the common functions and use of ParaView, for a more detailed description of the function and operation process please consult ParaView help documentation.

4.1 Start

4.1.1 Installation

- **Unix installation** - According to the installation instructions recommended to install to / usr / local or / opt directory.
- **Windows installation** - Follow the installation instructions to run the installer.

4.1.2 Start ParaView

There are several ways to implement ParaView, depending on whether it is running in a single processor mode or a distributed processor or client / server mode. Among them, the simplest execution mode is single processor mode. Here, we only introduce the single processor mode.

On Unix

If the installation path for ParaView is within the default search path of the current user ($ PATH or $ path), you can run ParaView with the following instructions:

$ paraview

If the path to ParaView does not join the current user's search path, you can add the path as the default search path, or use an absolute path to run ParaView.

For example, if the installation path is: / usr / local / bin, then the command in the following order:

$ / usr / local / bin / paraview

Will let ParaView recognize the command line parameters and allow you to view the list of available parameters for ParaView with the -help option.

On Windows

Click "All Programs -> ParaView-3.8.0 -> ParaView" on the Start menu or the shortcut on the desktop. (For information on how to run ParaView in a distributed manner, refer to the ParaView help documentation.)

ParaView Terms and Concepts

Here are some of the terms used in ParaView:
- **Object:** Simulates the entity state or behavior of the system.

- **Data objects:** Data abstraction. For example, when a .vtu file is loaded, the file passes through a data object representative.
- **Filter:** A process object that requires at least one input and produces an output. The filter includes: flat cut, take the filter, the threshold filter and so on.
- **Pipes:** In ParaView, process objects are interconnected to form a visual pipeline, each of the processes in the pipeline An object represents an operation on a data.

4.2 User interface

The ParaView 3.8 program interface shown in Figure 4.1, is composed of the following parts:

Menu Bar (menu bar)
System menu, all tasks can be done by clicking on a menu item.

Tool Bar (Toolbar)
Common tools for shortcuts collection.

Pipeline Browser
ParaView uses pipeline management data to read and operate the name of all data objects in the order of creation and in the form of tree structure. The sub data objects are listed below each parent data object in the order they were created.

Object Inspector (object properties)
The properties of the data objects in the pipe list, including the three attribute pages: Properties, Display, Information.

Display Area (Display Area)
The data visualization area of the object in the pipe list. Users can view, detect data, and can achieve the interaction of data manipulation. The area can be further subdivided into multiple different types of views and display the calculated result in different views.
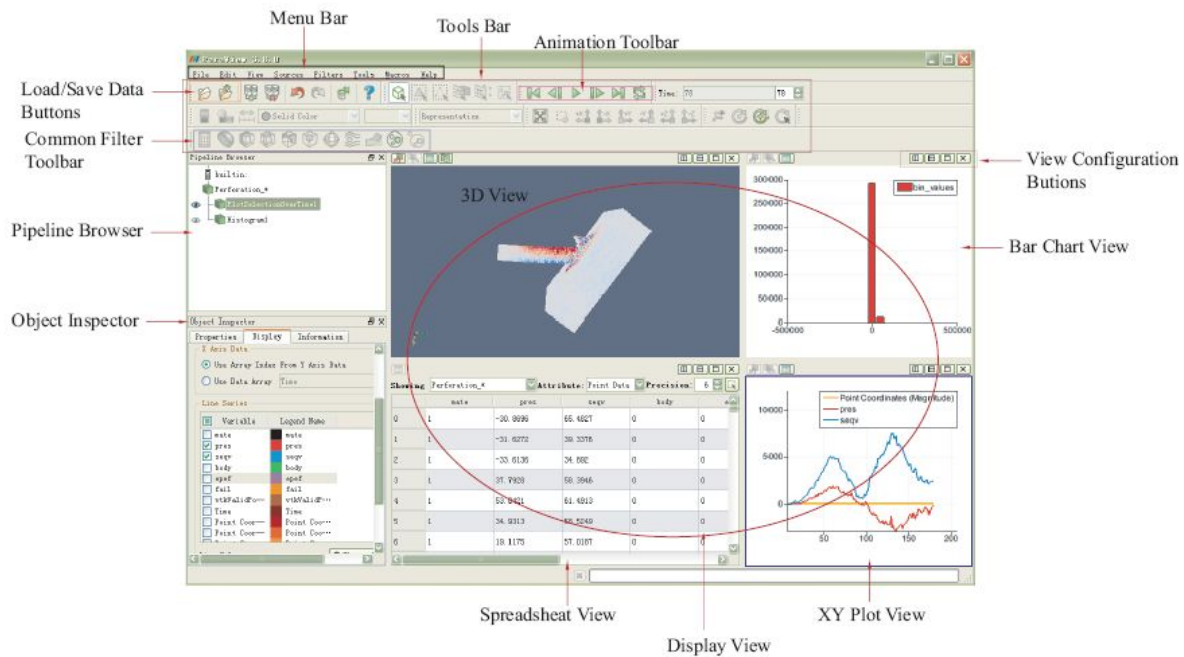
图 4.1: ParaView Interface

4.3 A simple example

Before we introduce the contents of each menu and the specific functions of the tool, we give a simple example of how to load a Peneblast dataset. There are two ways to get data:

1. to read data from a file or
2. to create data with a data source object.

We mainly use the first method to deal with PeneBlast system generated post-processing files (vtu and H5Part file). Suppose there is a post-processing file, *Perforation.vtu*. Click "Open" item in the "File" menu to open the file Folder, click the appropriate file, and then click the OK button, as shown in Figure 4.2. At this point, ParaView read the number

According to the file information, a data object named *Perforation* will appear in the PipeLine Browser. Now, Click the "Apply" button on the Properties folder of the Object Inspector to load all the data files and the 3D view is visible (the eye icon in the pipe list is black), as shown in Figure 4.3. The way data objects are drawn in the view area is controlled by the Representation Toolbar of the toolbar, Type data, we use Points to draw the graphics, as shown in Figure 4.4, select Points; display the physical quantity

through the tool

Column shortcuts Active Variable Controls drop-down menu to choose, we use the pressure physical volume, as shown in Figure 4.5

Show, select the variable pres. Click to display the color label for the pressure value, as shown in Figure 4.6.

Drag the mouse to move the 3D view: Press the left mouse button to move the mouse in the 3D View area to rotate the camera in the view area.

The rotation of the machine is around the center of the rotation; the middle mouse button to move the scene; right to use to zoom in and out of the scene.

4.4 Pipeline Browser

ParaView uses pipe management to load data objects or other data objects that are generated by other methods.

The sequence is listed in the form of a tree structure, as shown in Figure. The data objects in the pipe list are described as follows:

• Data objects are the body of system processing, and each data object has three property pages (collectively referred to as object properties).

Figure 4.2: ParaView opens the file

Figure 4.3: Loading the data file
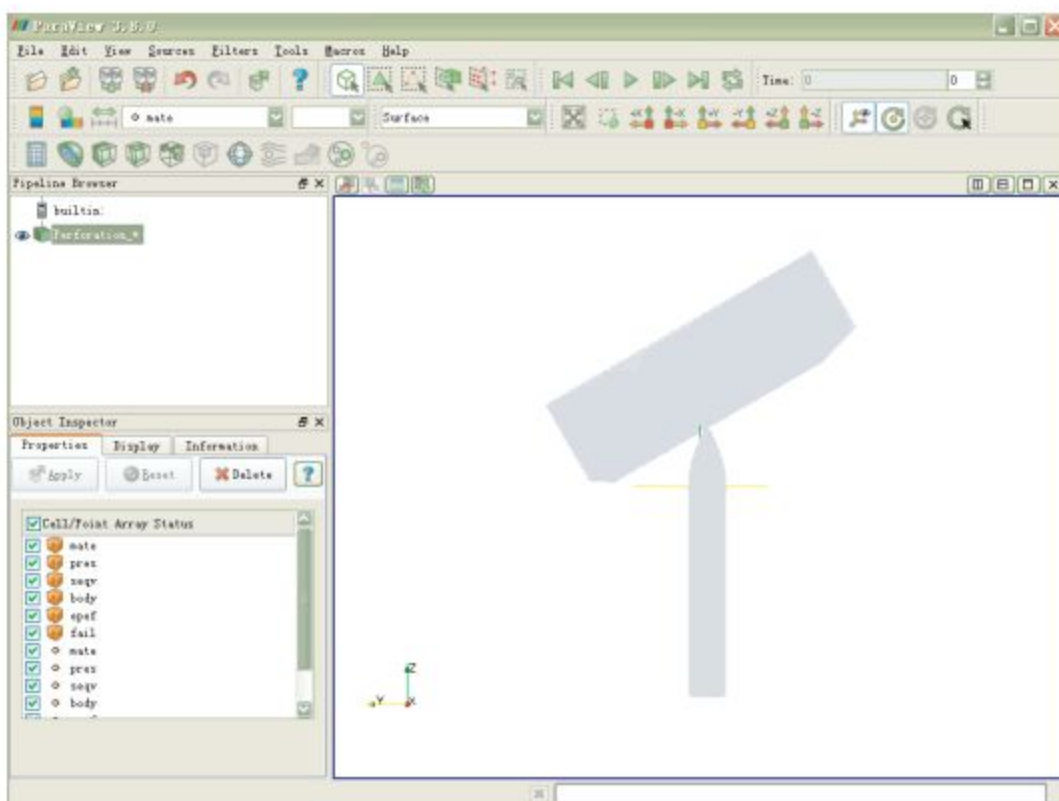
图 4.2: ParaView 打开文件

图 4.3: 加载数据文件

Figure 4.4: Select the drawing method

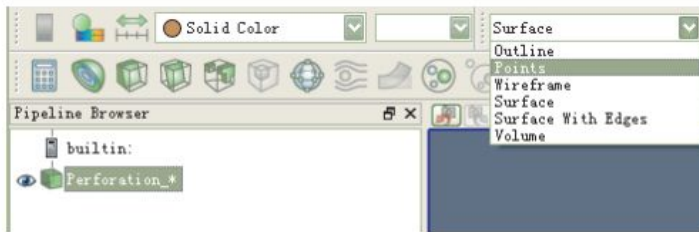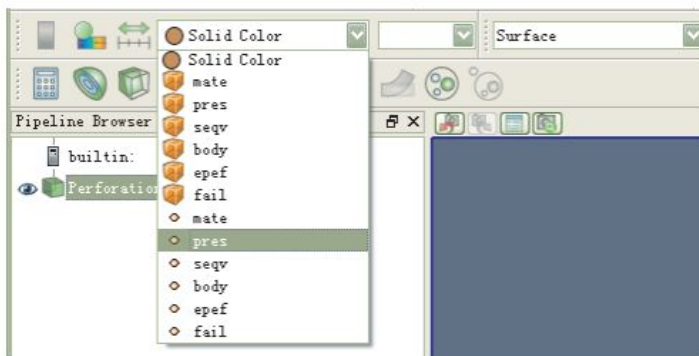Figure 4.5: Select the displayed variable

图 4.4: 选择绘制方式

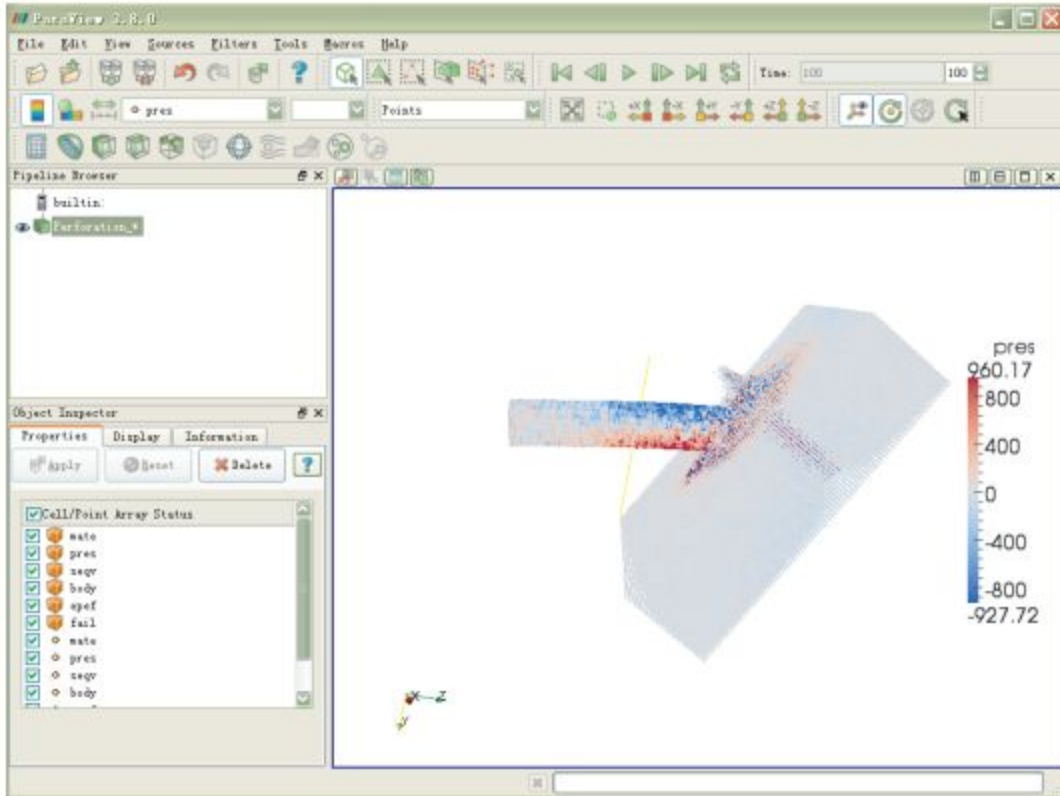图 4.5: 选择显示的变量

Figure 4.6: Pressure at step 100

图 4.6: 第 100 输出步的压力

• A granular or haplotype object dataset can be used as a parent, and a sub-dataset can be generated by a filter operation.

• For the same object, you can display its contents in different views in different ways or in different physical The amount to show.

For example, the object Performation includes the calculation of some of the particles at each time step, and so if The object is displayed in Spread Sheet View, then the list of the results of the mass at each moment is obtained.

• The object data must be activated when the data object in the pipeline is set or operated. **Pipeline**

A maximum of only one data object in the table is active.

• If there is an eye icon before the data object name, the icon controls whether the data object is visible in the current view.

The pipeline area responds to the following mouse operations:

• Left mouse button: Click the data object name to activate the data object; click the eye

icon in the current 3D View

Display the data object: gray indicates not displayed, black indicates display.

• Right mouse button: Click the data object name or eye icon, pop up the shortcut as shown in Figure? Click the blank area,

Pop-up shortcut as shown in Figure, only Open available.

The ParaView system supports the following sets of data sets:

1. Quality data

The collection of particle data, containing the coordinates of each particle and the calculation results, the particle data set can be created in the following way:

2. Grid-type data sets

A collection of grid data, a definition data containing a mesh, and a calculation result of a mesh node.

4.5 Object Inspector

Each data object has object properties, including three property pages: Properties, Diaplay, and Information. Correct

Like an attribute page is an important attribute of the data object that controls the data object: whether and how it is loaded into ParaView

The display of the area, some basic description of the data object. Use the left mouse button to click on the data object in the pipe list

The properties of the data object. Different property pages can be accessed by clicking on the tabs at the top of the window.

The content depends on the current data object type. The following describes the three property pages.

4.5.1 Properties

The property page consists of two parts: Operation buttons and Parameters Setting,

As shown in Figure 4.7.

Operation button

The operation buttons include three operations:

• **Apply:** Click to load the current data object according to the settings of the parameter area. The status changes from highlight to gray. when

When the parameter area is changed, the option is highlighted again.

• **Reset**: Click to restore the last parameter settings. This item is available if the parameter settings have changed;

• **Delete:** Deletes the current data object in the pipe list.

Parameter area

Different types of pipe objects, the contents of the parameter area is different, please refer to the specific ParaView help document. Here are three

Parameters for typical data objects: vtu / H5Part dataset, Clip (filter operation), PlotSelectionOverTime

(Filter operation).

• **vtu / H5Part file**: As shown in Figure 4.8, the parameter area lists the physical quantities carried by the data object. The system only loads

Check the physical quantity.

• **Clip**: As shown in Figure 4.8, the parameter area lists the cut geometry definition parameters;

• **PlotSelectionOverTime**: As shown in Figure 4.8, the parameter area lists the

selected list of particles and the Copy option.

When the contents of the parameter area settings have changed, then the operation button Apply, and Reset will be highlighted, click Apply

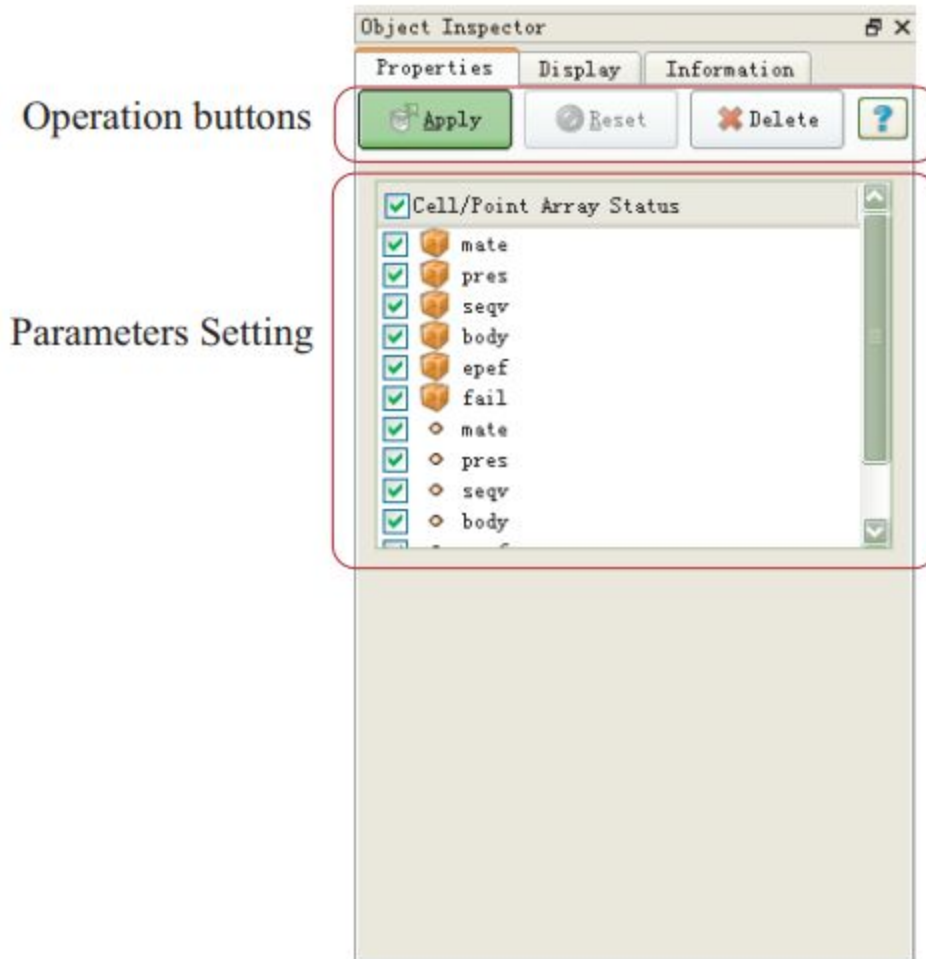Button settings will take effect, click the Reset button to restore the previous settings.

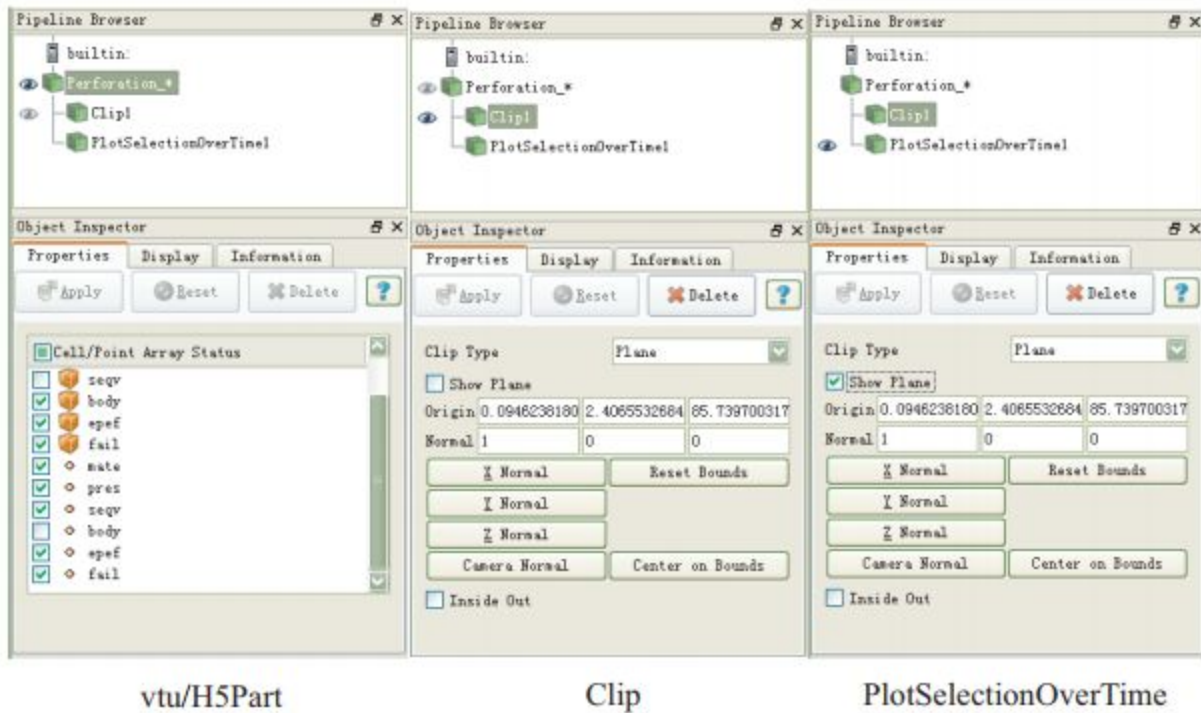图 4.7: 参数设置属性页

Figure 4.7: Parameter settings property page

图 4.8: 参数设置属性页

4.5.2 Display

The property page is used to set some parameters that control the pipe object when the view area is displayed, consisting of multiple sub-property boxes. Do not

The contents and number of the pipe sub-attribute box of the same type have changed. Only the data object and its view are selected

, The property page is available. The following describes the two typical data object Display property page: vtu / H5Part

Data set and PlotSelectionOverTime (filter operation).

vtu / H5Part file

The data object's property page consists of 10 sub-attribute boxes: View, Color, Slice, Annotation, Style, Edge

Style, Volume, Lighting, BackfaceStyle, Transformation. The following describes the Style property box,

Please refer to the ParaView help documentation for other information. The Style

property box is used to control the size of the displayed particles, as shown in Figure 4.9.

The meaning of each option is as follows:

• Representation

Function: Select the mapping method, pure point type data recommended use Points, containing the unit is selected Surface With

Edges.

• Interpolation

Function: Select the interpolation method for drawing the cloud.

• Material
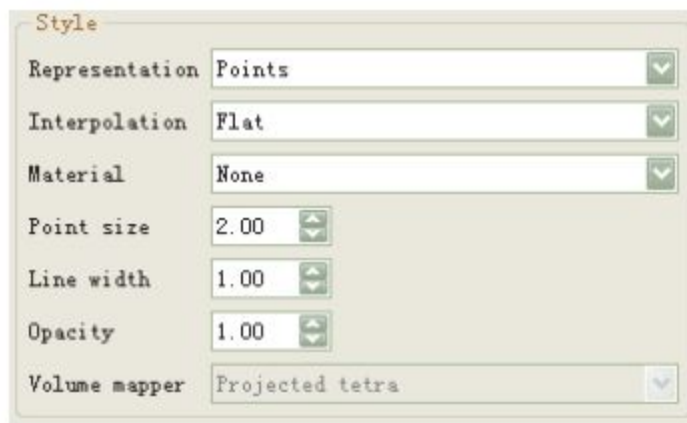
Function: Please refer to the ParaView help documentation

图 4.9: Diaplay Style 属性内容

• Point size

Function: Adjust the displayed particle size.

• Line width

Function: Adjust the width of the display unit edge.

• Opacity

Function: Adjust transparency.

• Volume mapper

Function: Please refer to the ParaView help documentation.

PlotSelectionOverTime

The attribute page of the data object consists of eight sub-attributes: Select Block, Attribute Mode, X Axis Data, Line

Series, Line Color, Line Thickness, Line Style, Marker Style, as shown. The following is

a brief introduction

Under Select Block, X Axis Date, Line Series property box. Please refer to the ParaView help documentation for other information.

• Select Block: The list of observed particle IDs, the time curve for the selected physical quantities of the particles in XY

Plot View show. The place can only be selected.

• X Axis Data: Select the contents of the x-axis

• Line Series: A list of physical quantities.

4.5.3 Information

The property page displays some statistics for the pipe object. For different types of data objects, the contents of the property page will be

Different. Here we only give the introduction of the imported particle data created by the relevant content of the pipeline object. Other please

Refer to the ParaView help documentation. For a pipe object with particle / unit data, the properties page is shown in Figure 4.11.
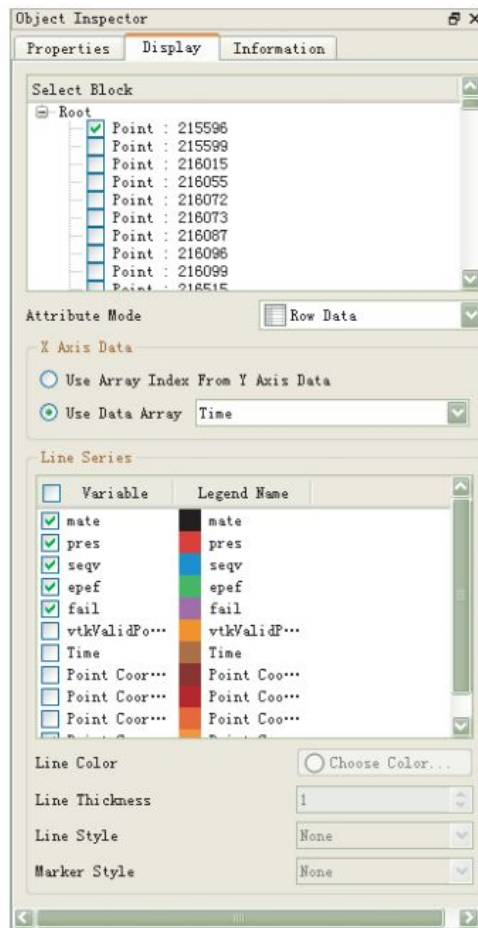
Specifically includes five items, as follows:

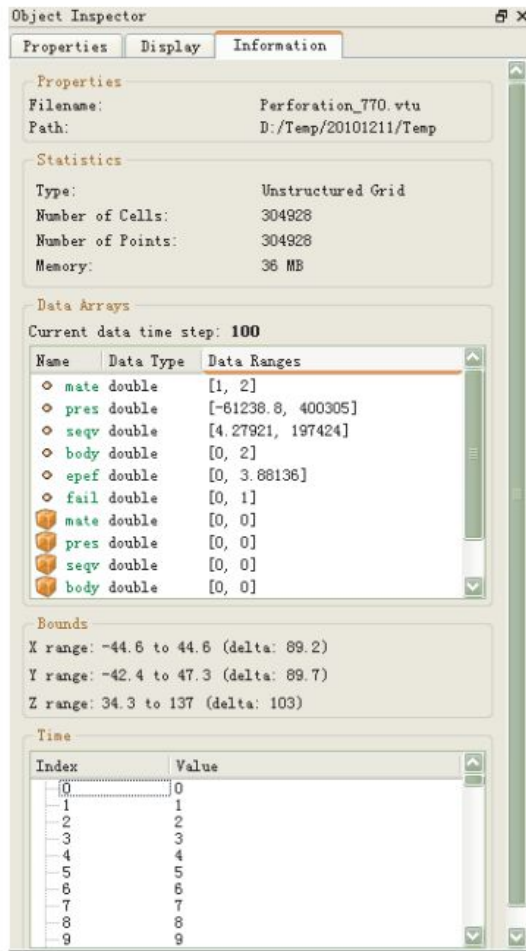图 4.10: PlotSelectionOverTime 数据对象的 Diaplay 属性内容

图 4.11: Information 属性内容

- Properties: Displays the filename of the current frame of the pipe object and the path where the file is saved.
- Statistics: Displays some statistics for the pipe object, such as data type, number of cells, number of points, file size.
- Data Arrays Function: Displays the data type of the physical quantity in the current frame and the range of the physical value
- Bounds Function: Displays the boundaries of the material area in the current frame
- Time function: Refer to the ParaView help file

4.6 drop-down menu

4.6.1 File menu
The File menu is mainly used to open and save ParaView various types of files.
- Open
  Function: Open the data file. PeneBlast's results are stored in H5Part or vtu's data format In the H5Part as a file, and vtu file is composed of multiple files, the file corresponds to an output step.ParaView can load a file, you can also load a series of vtu files, the formation of animation.

  - Process: Click this option to bring up the file selection dialog (shown in Figure 4.2). Select the file you want to open -> Object
    Browser to add the file name named object -> Object Inspector show Properites property page,
    Lists the result variable names contained in the data file, and the user can select the results to be processed.
    The Apply button in the page, the physical model of the first time step will be displayed in the current view (if the current view
    Not the 3D View view, it is not displayed).

- Recent Files feature: Open recently opened files.

  - Process: Click this option to list the recently opened list of files in the form of a submenu under this menu item.
    Select the file name in the table, the system according to the file extension to determine the open mode.

- Save State

  - Function: Save the current GUI status and all data sets of the ParaView to a file (with the extension sta),
    With Load State, you can restore the ParaView to the saved state (including the GUI interface and all datasets).
- Load State
  Function: read the saved ParaView state from the sta file and restore the

ParaView to that state. In the case of

- Save Screenshot
  Function: Saves a screenshot of the current view.

  - Process: Click this option to pop up the file selection dialog box to select the file type to be saved (png, tif, bmp, jpg, pdf)
    -> Save the snapshot of the current view in the specified format to the file. Save Screenshot can not be used with Spreadsheet
    View view (the menu item will be grayed out).

- Save Animation
  Function: save the animation.
  Process: Click this option to bring up the Animation Settings dialog box, as shown in Figure 4.12. In the dialog box, click Save Animaiton, the pop-up dialog box, as shown in Figure 4.13. You can choose to save the format
  (avi, jpg, png, pdf, etc.), Paul
  Save the path and edit the file name, the time display area of the snapshot is saved in the specified format. Only when it has been opened
  Post-processing data files (vtu / H5Part format) are available. If you choose avi format, save as an animation, if selected
  Select the picture format, then save each frame as a picture.
- Exit
  Function: Exit the program.
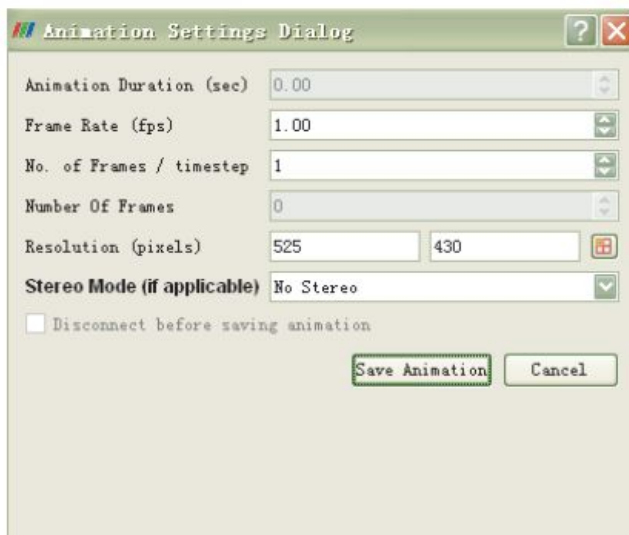 Process: The confirmation dialog box pops up -> Ok or Cancel.
- Load State
  Function: read the saved ParaView state from the sta file and restore the ParaView to that state. In the case of
- Save Screenshot
  Function: Saves a screenshot of the current view.
  Process: Click this option to pop up the file selection dialog box to select the file type to be saved (png, tif, bmp, jpg, pdf)
  -> Save the snapshot of the current view in the specified format to the file. Save Screenshot can not be used with Spreadsheet
  View view (the menu item will be grayed out).
- Save Animation
  Function: save the animation.
  Process: Click this option to bring up the Animation Settings dialog box, as

shown in Figure 4.12. In the dialog box, click Save Animaiton, the pop-up dialog box, as shown in Figure 4.13. You can choose to save the format (avi, jpg, png, pdf, etc.), Paul

Save the path and edit the file name, the time display area of the snapshot is saved in the specified format. Only when it has been opened

Post-processing data files (vtu / H5Part format) are available. If you choose avi format, save as an animation, if selected

Select the picture format, then save each frame as a picture.

- Exit
  Function: Exit the program.
  Process: The confirmation dialog box pops up -> Ok or Cancel.

图 4.12: 动画设置对话框

- Setting
  Function: Set the global parameters of the program. Click this option to bring up the Settings dialog box, as shown in Figure 4.14 (on the left
  The setting of each category, the right part of the settings for the details) shown.
- ViewSetting
  Function: lists the contents of the current view can be changed, the corresponding content will be based on different types of views are different.

4.6.3 View menu

This menu is used to customize the shortcuts on the ParaView toolbar, such as animation controls, pipe lists, object properties pages
Wait.

- ToolBars
  Function: control the tool bar shows the shortcut, click on its subordinate menu of an item to switch the item in the tool
  Column shows / hides the status as shown in Figure 4.15.

- Annimation View
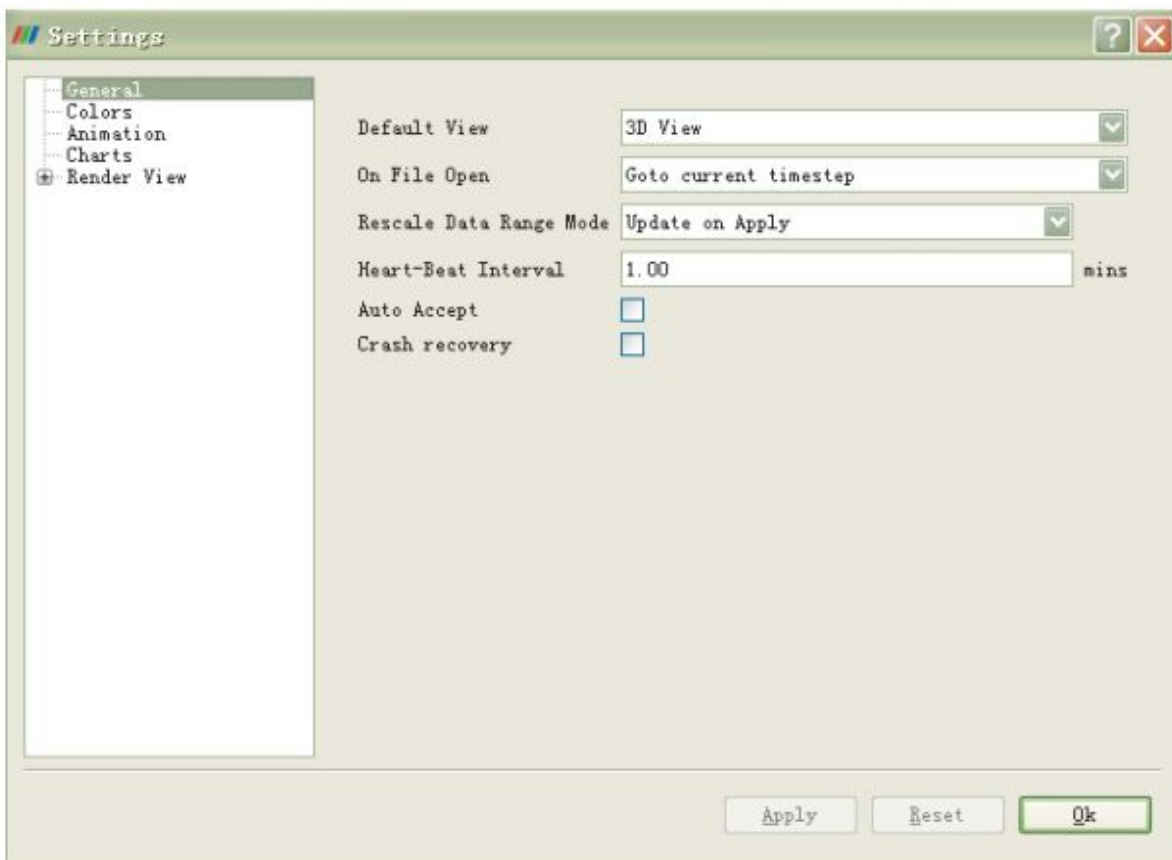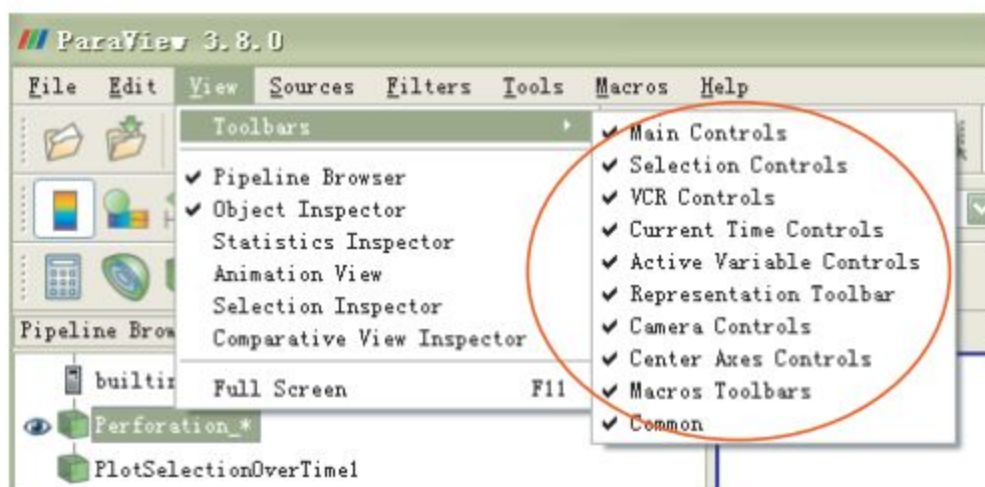 Function: Controls whether the animation playback status control bar is displayed on the interface.

图 4.14: 全局设置对话框 Global Settings dialog box

图 4.15: 工具栏设置 **Toolbar settings**

Chapter 5 PeneBlast Getting Started Guide

In order to facilitate learning to use the PeneBlast system as soon as possible, this chapter through a few examples of the software in detail

The process, in which the modeling, solution and post-processing of the oblique invasion are described. In addition, it is provided

Taylor rod, two-dimensional TNT detonation, broken, steel ball penetration of concrete, Gurney flyer, jitter jets, they

From the study description, task definition, material definition, component settings, grid settings, contact settings, detonation settings,

Set the output settings, check the modified input file and calculate the monitoring of several parts of the PeneBlast system in the specific instances.

5.1 Guide to Diagonal Penetration Model

5.1.1 Model Overview

The example simulates a kinetic energy bomb with an initial velocity of 575 m / s penetrating an aluminum plate for the study of kinetic energ performance, more concerned about the kinetic energy bullets before and after the speed of armor. The kinetic energy is a cylinder with an oval warhead, the material is steel,
The target material is aluminum. The corresponding size of the study model is:
1. The diameter of the cylindrical part of the projectile is 12mm, the length is 89mm, the radius of the oval warhead is 6mm, the appearance
   Surface radius of 18mm;
2. The thickness of the target is 26mm, 96mm long and 100mm wide;
3. Projectile with a target angle of 30 degrees.

In this example, the unit system uses mm ms g

The model diagram is shown in Figure 5.1. :

(No picture in original manual for fig 5.1*)

Material parameters
According to the test situation, under normal circumstances, the wear of the missile is relatively less, and the damage of the target is very serious. So in the calculation
The material model used for each object in the simulation is shown below:
1. The elastomeric model of the elastomer is in the MPM3DPP program, and the IsoHarden material
The parameter value is shown in Fig.
2. The Johnson-Cook model is used for the target model, the state equation is Gruneisen's state equation, the failure is maximum, etc.
The plastic strain model PlaStrain, the corresponding parameter values are shown in Fig.
Modeling method
Specific modeling process has the following key considerations:
1. The model has a symmetry shape, in order to improve the computational efficiency, it is desirable to model half of the model;
2. As the pre-processing can not be generated once the missile body, so the half of the missile model to be generated separately, that is, the first half

Cylindrical body, and then the establishment of half of the oval body, thus the two body synthetic elastomer model;

3. Since it is not possible to directly generate a tilted rectangular parallelepiped, it is necessary to define a translation plane or work plane after the translation, and then The target plane is established in this work plane (local coordinate system);

4. The study should enable the contact algorithm, so to establish two components, the above-mentioned discrete parts are classified into the corresponding components;

5. In this example, the background grid size is taken as 2 × 2 × 2mm and the particle distance is 1mm.

Summary of steps

The flow of the study is divided into three parts, the first part is pre-processing, used to generate MPM3DPP program can read

* .xmp file; the second part with the Peneblast system call MPM3DPP program to read the corresponding data file and

Line calculation; the third part of the post-processing, the use of Paraview generated data files for a variety of data processing. The following Jane

To list the steps.

Pretreatment

1. Specify the working path, task name, unit system;
2. Define the material model
3. Establish discrete models
4. Define the background grid
5. Define the contact algorithm
6. Specify the output options
7. Specify the Solver option
8. output definition file

Solve the calculation

1. set the execution control
2. Perform the calculation

Postprocessing

1. Read the results of the solution
2. animated presentation
3. Select the particle
4. slices, cut, view the internal particles

5.1.2 Pretreatment

Step 1 Specify the working path, task name, and unit system
The function of the pre-processing is to define the input file for the MPM3DPP program to read, so it is generally necessary to specify a file for the file
Save the path, the file name, and some simple descriptions of the study; you also need to specify the unit used to solve the problem
system. Open the Peneblast program and edit the Job Properties of Job1 in the Job Browser, as shown in Figure 5.2.
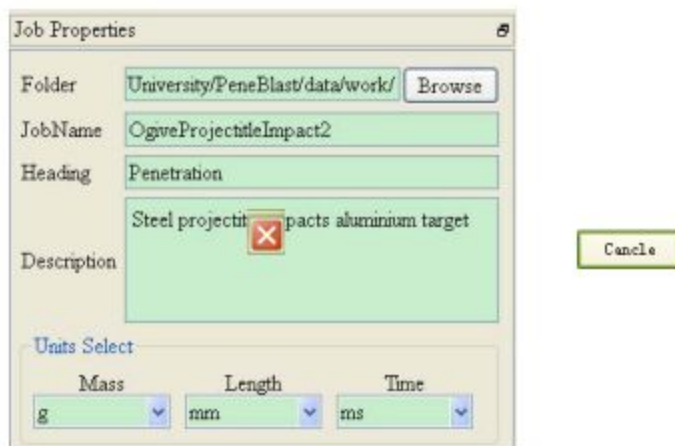
图 5.2: 工作属性

Step 2 Define the material model
In this paper, the definition of the study of all the material model, the material model is defined in a variety of ways, can be used directly
Materials have the material, modify the material in the material library as a new material or directly create a new material. In this study only
Using two kinds of material model, in this new way to define, the following one by one.

1. Models-> Material pops up as shown in Figure 5.3;
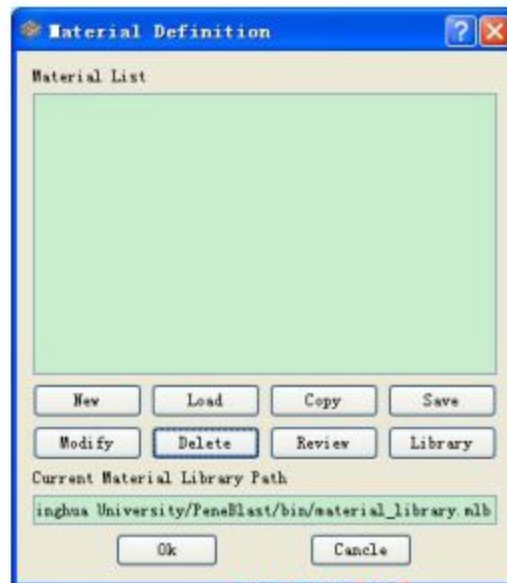
图 5.3: Material Definition dialog box

2. Click New to display the dialog box shown in Figure 5.4.
    Here you can specify the material name, density, intensity model, state equation and failure parameters such as value. Respectively, with the corresponding under
    Pull the menu to find the use of the material model, fill in the corresponding material parameters can be completed, click the button after filling
    The parameter value should be read, exit the upper left corner, cancel the current settings click button. For this example
    The two material models are filled in Figure 5.5 and Figure 5.6 below.

Step 3 Defines the discrete model

For this example, you need to create two components to enable the touch algorithm. The missile is set as the component projectile, the target is set to
The component target, each discrete model, needs to be generated under the specified component. The establishment of the missile body is slightly complicated due to the projectile
For the oval shells, need to establish a separate body and warheads. As long as the warheads and bones of the discrete particles at the recent interval of not more than a net

The size of the grid is considered to be one of the two. Because of the symmetry of the problem, it is only necessary to build half the model.

1. Build the component projectile
   Can be built through the toolbar Models-> Create Component or right-click PreDataSet to select New
   The components are shown in Figures 5.7 (a) and 5.7 (b), respectively. In the PreDataSet appears component1, double-click component1
   Modify it to projectitle.
2. Create bones in the component projectile Guidelines for Diagonal Invasion. The bomb is a cylinder, where half of the model is created.
   a. Select projectitle
   b. Click Models-> Create Cylinder, or right click PreDataSet to select New Cylinder and Create a cylinder, as shown in Figure 5.8 (a) and Figure 5.8 (b). After the interface on the left side of the interface shown in Figure 5.9, is divided into three parts. The first part of the Properties property page Used to fill in the geometric information, the second part of the Load Setting property page is used to set the load parameters, the third part of the Display
      The page is used to set the display parameters, the fourth part of the Discretization property page is used to set the discrete parameters.
   c. fill in the Properties page attribute value, as shown in Figure 5.9 below. in the size box to fill out the geometric information, the specific meaning of the manual;
   d. in the Velocity box to specify the initial speed of the object, this time the target body in the top of the body, so as shown in Figure 5.9;
   e. in material Specifies that the object's material model is Steel;
   f. Click the Discretization button to make discrete settings, which are mainly used for particle spacing settings, as follows In Figure 5.10; the body model is shown in Figure 5.11.
3. Under the component projectile, establish a semi-oval body discrete model.
   a. with the left mouse button click projectitle, select the component;
   b. Click Models-> Create Ogive, or right click PreDataSet to select New Cylinder to create Oval, as shown in Figure 5.12 (a) and Figure 5.12 (b).
   c. fill in the Properties page attribute value, as shown in Figure 5.13 below.
   d. Click the Discretization button to make a discrete property page for discrete parameter settings, as shown in Figure 5.10.

The same situation. The warhead model established is shown in Figure 5.14

4. Left click on the component projectile to see half of the model of the complete projectile on the right side of the window, as in Fig 5.15 (trivial)

5. Build component

Since this example simulates oblique penetration, the target plane needs to be used in the work plane, first in the PreDataSet property page

WorkPlane defines the work plane, as shown in Figure 5.16. Define the work plane WorkPlane1, see Table 5.1 for specific parameters

See the MPM3D User's Manual for parameter details.

6. Models-> Create Component, or right-click PreDataSet to select New to create the target component

Component2 appears under PreDataSet and double-click component2 to change it to target.

6. Build the target model under the component target

6.1 selected components target:

6.2 Click Models-> Create Block or right click on to select New Block to create the box,

In this case, the property page of Block is displayed on the left side of the window, as shown in Figure 5.17.

6.3 Click Properties to enter the property page for geometric parameter setting and initial speed setting, as shown in Figure 5.17.

6.3.1 fill in the Block size box geometry information;

6.3.2 Select WorkPlane1 in the WorkPlane drop-down box;

6.3.3 Select the Aluminum material in the material box;

6.3.4 Velocity check box is not checked;

6.4 Click the Discretization tab to enter the parameter page to set the parameter discrete parameter, where dp is 1, with

The discrete process is automatically calculated. The settings are shown in Figure 5.18.The resulting target is shown in Figure 5.19.

At this point, the discrete model has been completed, click PreDataBase to see the overall model shown in Figure 5.20, the following into

Into the PreDataBase property page settings.
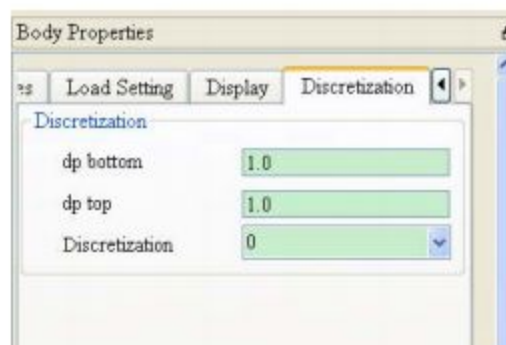
7.
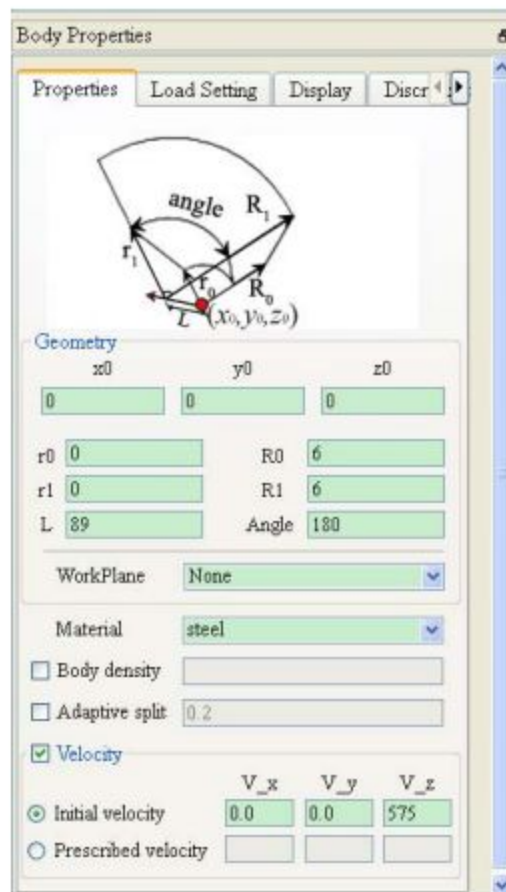
图 5.9: Properties 属性页设置

图 5.10: 弹身离散设置

表 5.1: 工作平面参数

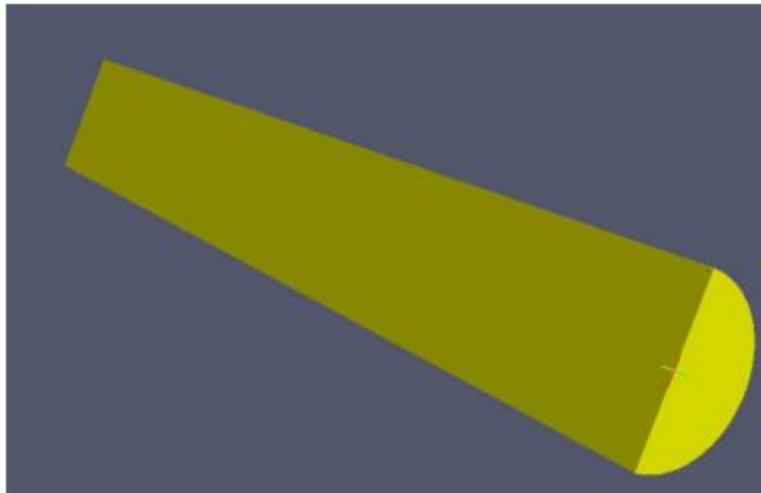| Name	r0x	r0y	r0z	ex	ey	ez	all	a21	a31
WorkPlane1	90	0	70	-1.732	0	1	1	0	1.732

图 5.11: 弹身模型

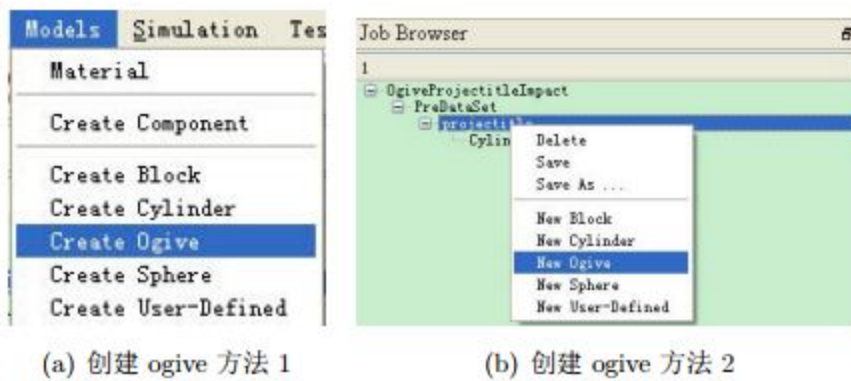(a) 创建 ogive 方法 1   (b) 创建 ogive 方法 2
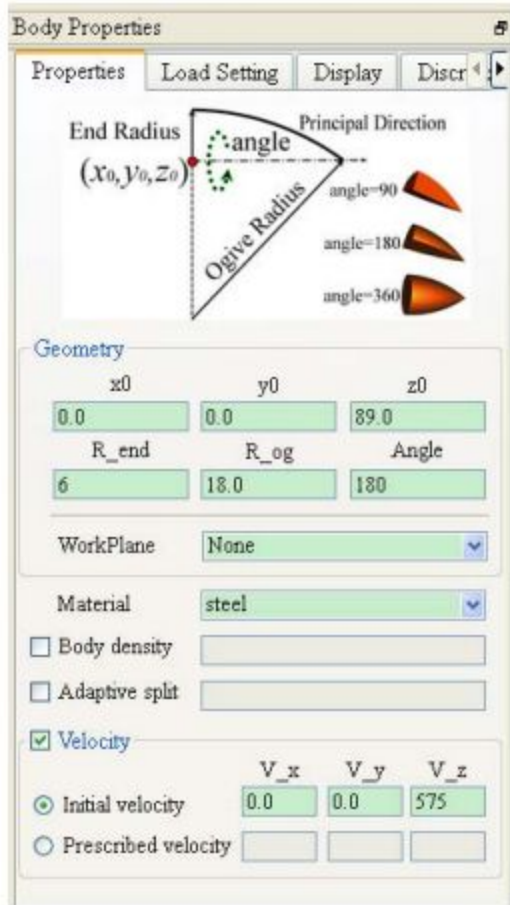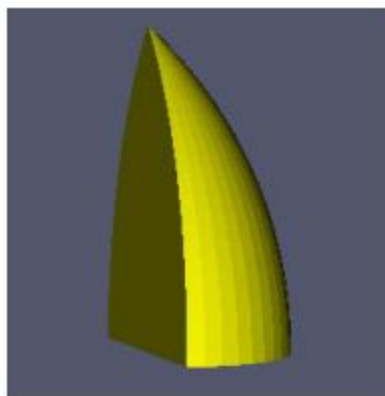
图 5.12: 新建卵形体
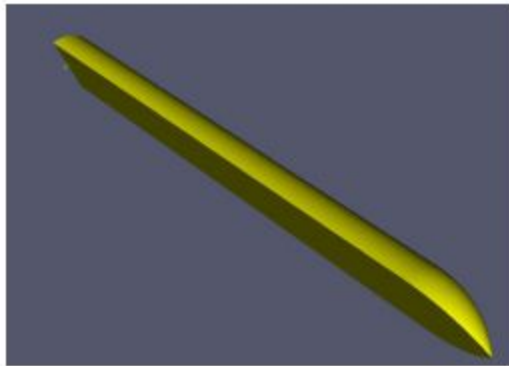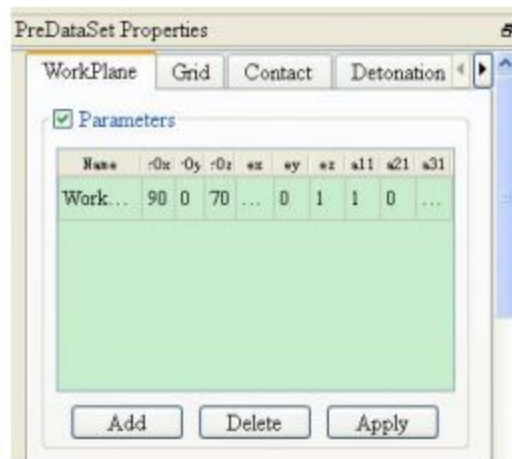
图 5.13: ogive 属性页设置

图 5.14: 弹头模型

图 5.15: 弹体模型

图 5.16: 定义工作平面

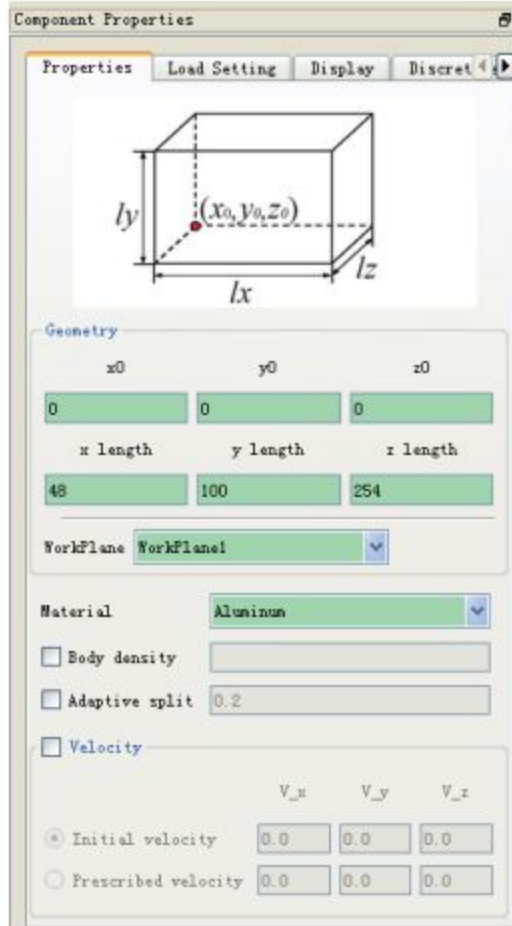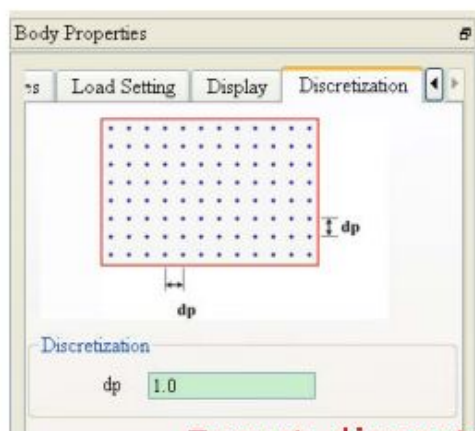**5.15 Projectile model; 5.16 Define the work plane**

图 5.17: Block 属性页设置

5.17 Property Page Settings

Target discrete
settings
图 5.18: 靶体离散设置

图 5.19: 靶体模型 Target
model
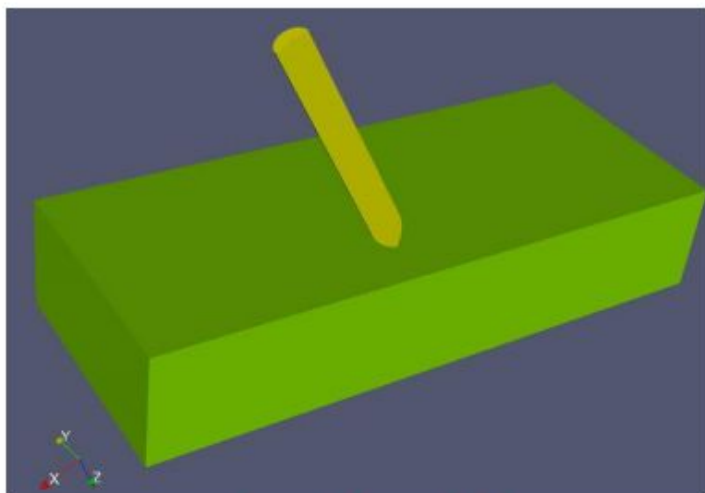
Overall model
图 5.20: 整体模型示意图

98

Step 4 Set the background grid

In principle, the background mesh as a range of motion that should completely cover the object within the simulated physical time. If used to move
Dynamic grid, depending on the circumstances at least cover the initial time occupied by the object space position. This example, in this dynamic grid,
The mesh area covers the motion range of the object. In this case, should also set the corresponding boundary conditions, and whether to use GIMP shape function. details as follows.
4.1. Click the PreDataSet on the left side of the window, where Job Properties can be set for the Grid,
Click on the Grid to enter the background grid settings page, according to the specific circumstances of this study, the corresponding border size and border conditions with The body settings are shown in Figure 5.21.

4.2 Set the background mesh discrete size on the Grid property page, as shown in Table 5.2.
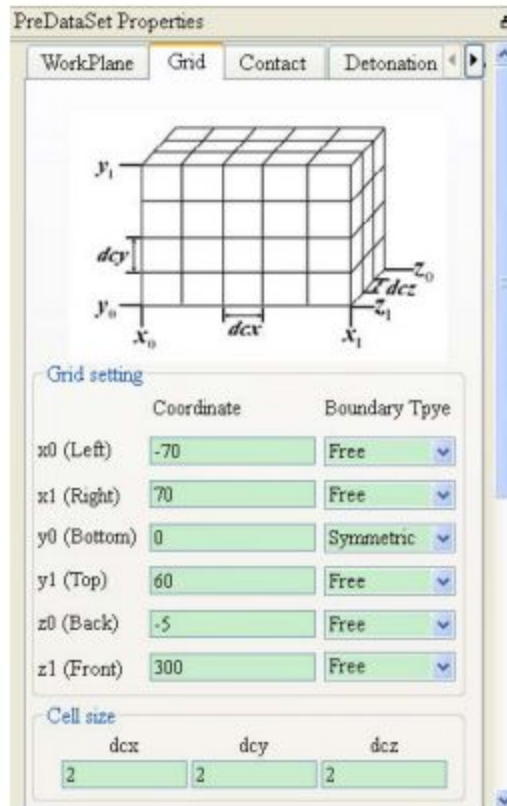
图 5.21: grid **Discrete Settings**

4.2 在 Grid 属性页设置背景网格离散尺寸，如表5.2。

Table 表 5.2: Grid setting

	Coordinate	Boundary Type
x0(Left)	-70	Free
x1(Right)	70	Free
y0(Bottom)	0	Symmetric
y1(Top)	60	Free
z0(Back)	-5	Free
z1(Front)	300	Free

4.3 Set the grid spacing to 2 × 2 × 2 on the Grid property page

4.4 Set the dynamic grid on the Grid property page to save memory when calculating, as shown in Figure 5.22
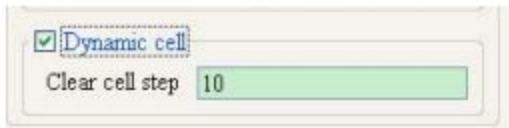
图 5.22: 动态网格设置

At this point, the background grid of the relevant parameters according to the specific circumstances of this study have been successfully set up.

Step 5 Turn on the contact algorithm

This example establishes two components, with the touch-enabled algorithm. So the pre-processing need to open the contact function; the current example
Does not consider friction.

Click Contact to enter the relevant parameter settings page, as shown in Figure 5.23.
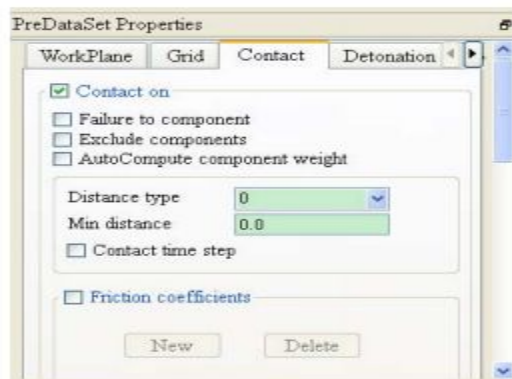
图 5.23: Contact settings

In addition, manually set the weight of the missile and target contact algorithm calculation, respectively, select the components projectitle and target, in the respective Component Properties under the settings, where the weight of the body is bigger, the missile body and the target
The weight settings are shown in Figure 5.24 (a) and Figure 5.24 (b), respectively.

Step 6 Set the output options

In this case, set the physical quantity to be observed in the post-processing file, the number of output files, and the time of the console monitoring Separated
6.1 Click the Output button to enter the corresponding property page, as shown in Figure 5.25;
6.2 Set the console output value. For the current example, the setting is shown in Figure 5.25.
6.3 Select the physical quantity to be observed, and check it in the box in front of it. For the example, see Figure 5.26.
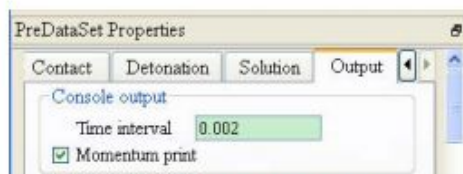6.4 Select the output file format, and the output time interval, for the study set Figure 5.26;
At this point, the output options for this study are set.
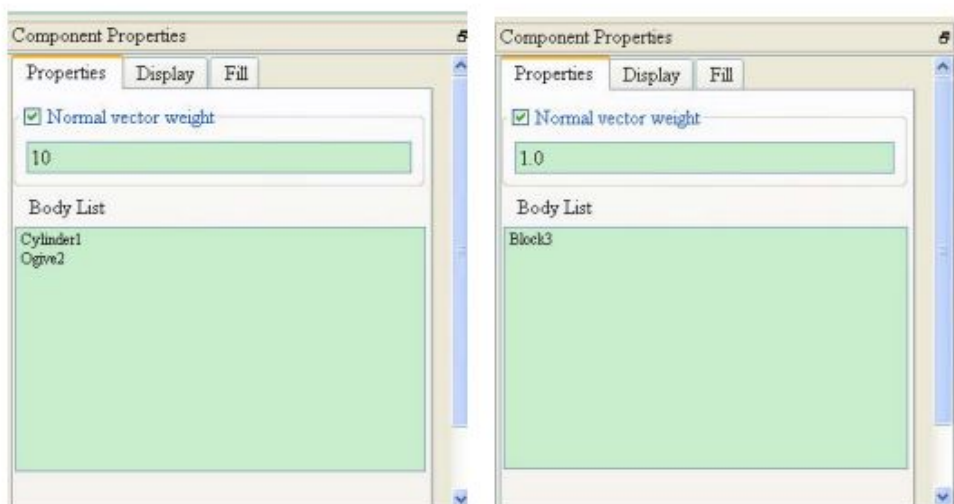
(a) 弹体权重设置 (b) 靶体权重设置

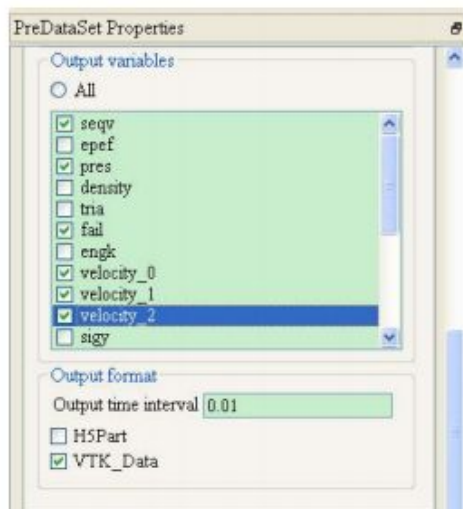图 5.24:Component contact weight setting

Output settings

图 5.25: 输出设置

Output settings (contd)

图 5.26: 输出设置

Step 7 Set the Solver option

Here, we set some settings for solving, and the application of gravity field.

7.1 Click the Solution button to enter the property page, as shown in Figure 5.27;
7.2 Set the simulated physical time End time to 0.8 and the time step factor is 0.2
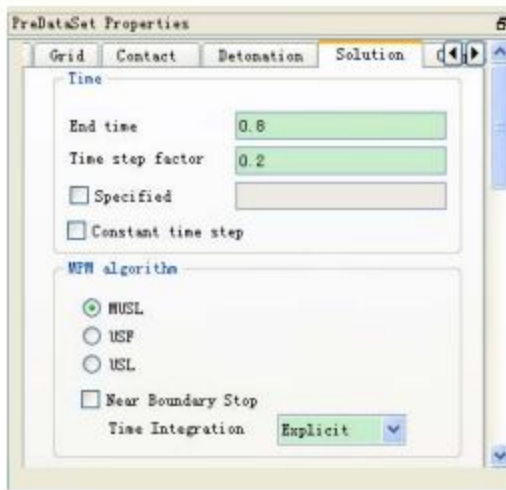The settings of this study are shown in Figure 5.27;

图 5.27:Solve Settings

7.3 Set the MPM algorithm, for the study, select the MUSL format;
At this point, the options for the solution are set up.

Step 8 Save the input file

The above seven steps constitute the basic steps of the pre-processing of this example,
and the work is saved to the MPM3DPP program.
Enough to read the suffix for the xmp file.
Click File-> Save Job, save the file to the current working path, as shown in Figure 5.28,
generate OgiveProjectitleImpact.xmp
Enter the file. Also you can right-click the task in the Job Browser and select Save to do
this.

图 5.29: Open the input file

图 5.30: Edit the input file

Click Simulation-> check Input Data to check the input file, pop-up dialog box, as shown in Figure 5.31

You Must check xmp through format check prior to solving.

5.1.3 Solving calculations

Step 10- perform the calculation

1. Simulation-> Real time monitor Turns on real-time monitoring.
2. Toolbar-> Start, perform the calculation, calculate the monitoring screen as shown in Figure 5.32.
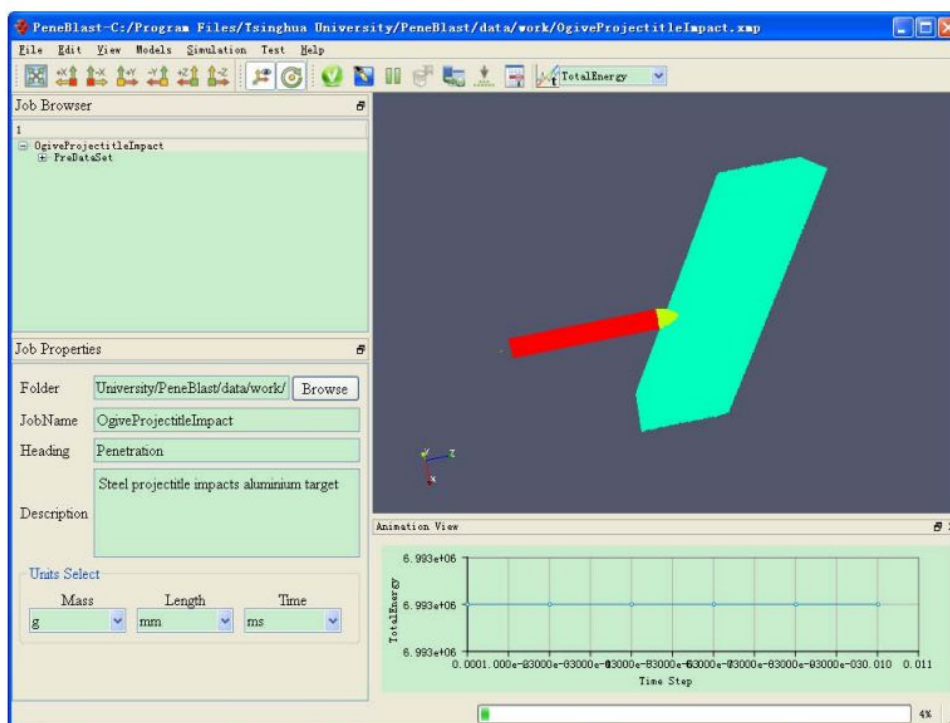

图 5.32: Calculation monitoring

3. During the calculation, you can pause, terminate the calculation process and change the XY Plot variable.

4. After the calculation process is paused, click Save Restart Files to save the restart status file.

5. If you continue to calculate from the interrupt state, in the option interface to load the restart state file, if you want to restart the command file(* .rmp), set together.

Step 11 Read the solution

Use Paraview software for post-processing, read and solve the output file
Menu-> File-> Open Load the resulting file, as shown in Figure 5.33.
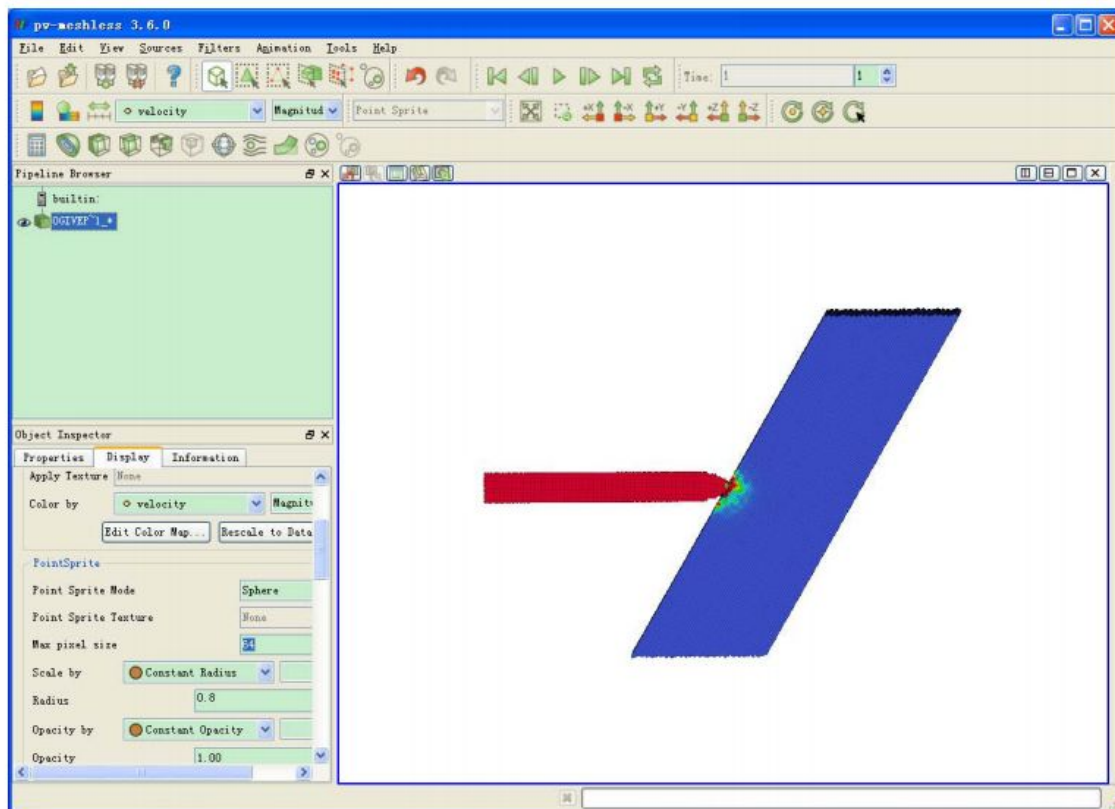

图 5.33: Load the result file

1. Press the work button on the toolbar to view the tip of the projectile, as shown in Figure 5.34.
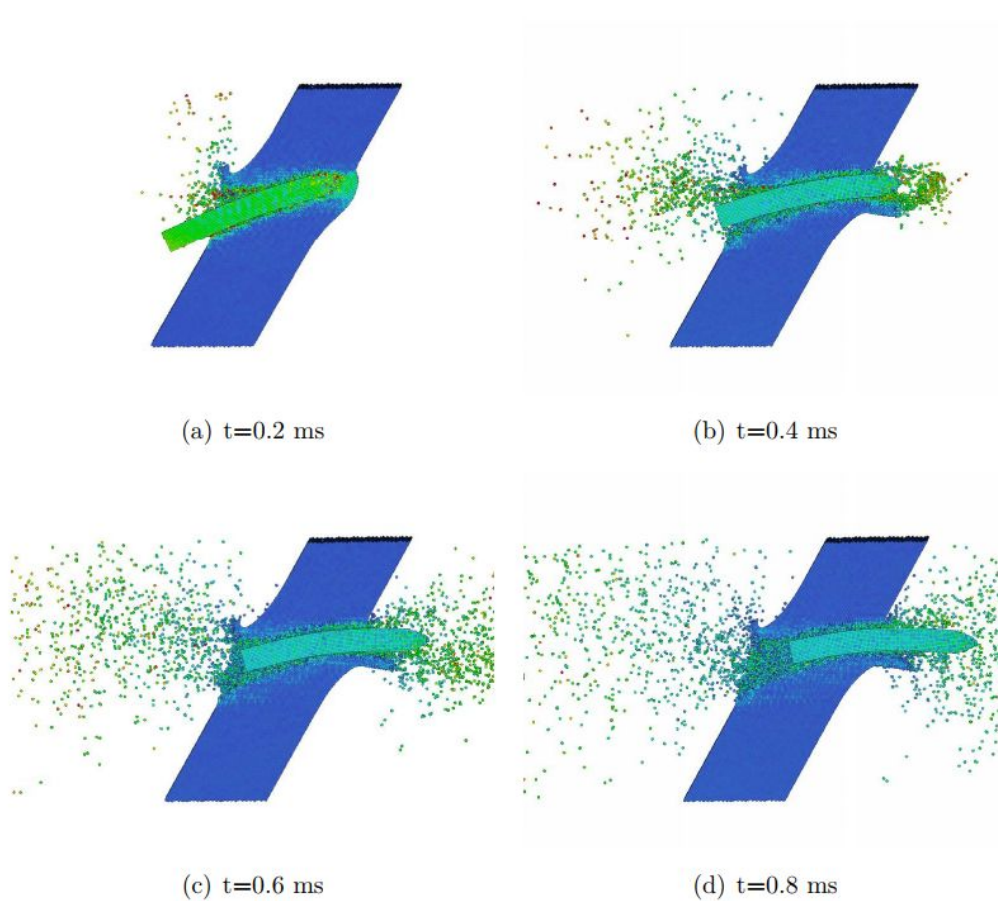

(a) t=0.2 ms   (b) t=0.4 ms

(c) t=0.6 ms   (d) t=0.8 ms

图 5.34: Penetration process

2. In the Variable check box, select fail to check the URL failure status, as shown in Figure 5.35.

3. Select the point using Select Points Through. Then select Extract Delection from the Filter menu and click on the Properties page

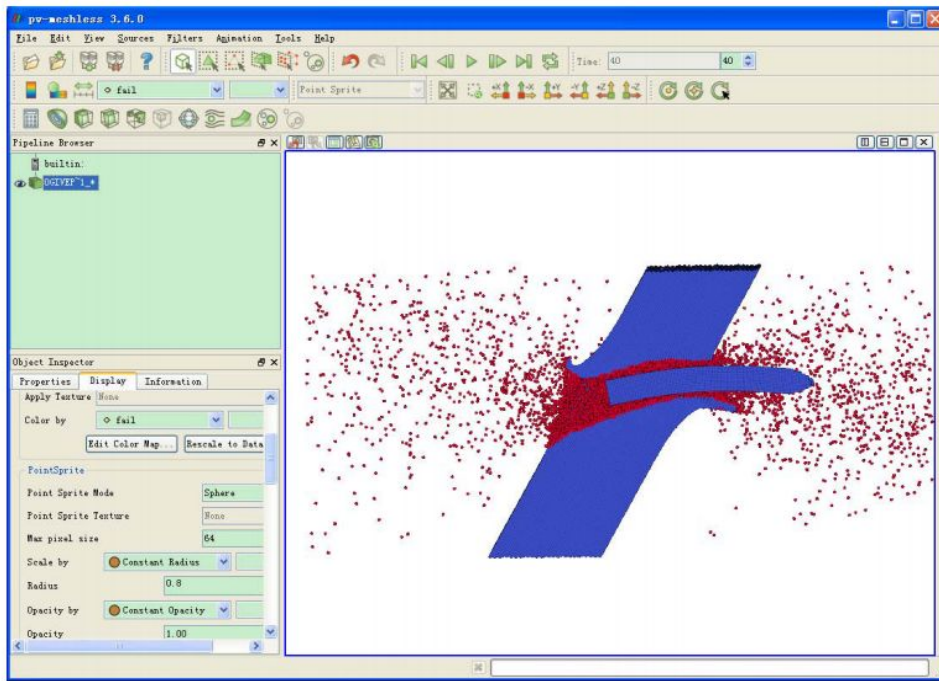Copy Active Selection to get the selected particle.

图 5.35: Failure particle distribution

3. Select the point using Select Points Through. Then select Extract Delection from the Filter menu and click on the Properties page
Copy Active Selection to get the selected particle.

The Spreadsheet view is an important tool for connecting data selection and numerical analysis. It allows the user to view the scalar domain
The actual value of the selection mechanism to help users identify the value of interest. Use or divide the view into two halves.

In the new view, click the "Spreadsheet View" button to monitor the selected data set, click to make the number
As shown in Figure 5.36, the operation examples are shown in Figure 5.36.

4. Paraview has the ability to draw time series field graphs. Drawing operations are based on data selection Row.
First select the data set, click the menu Filters-> Data Analysis-> Plot Selection Over Time,
In the Object Inspector, click "Copy Active Selection", and finally click to get the selected
The time-history curve of the particle-dependent variables, as shown in Figure 5.37.
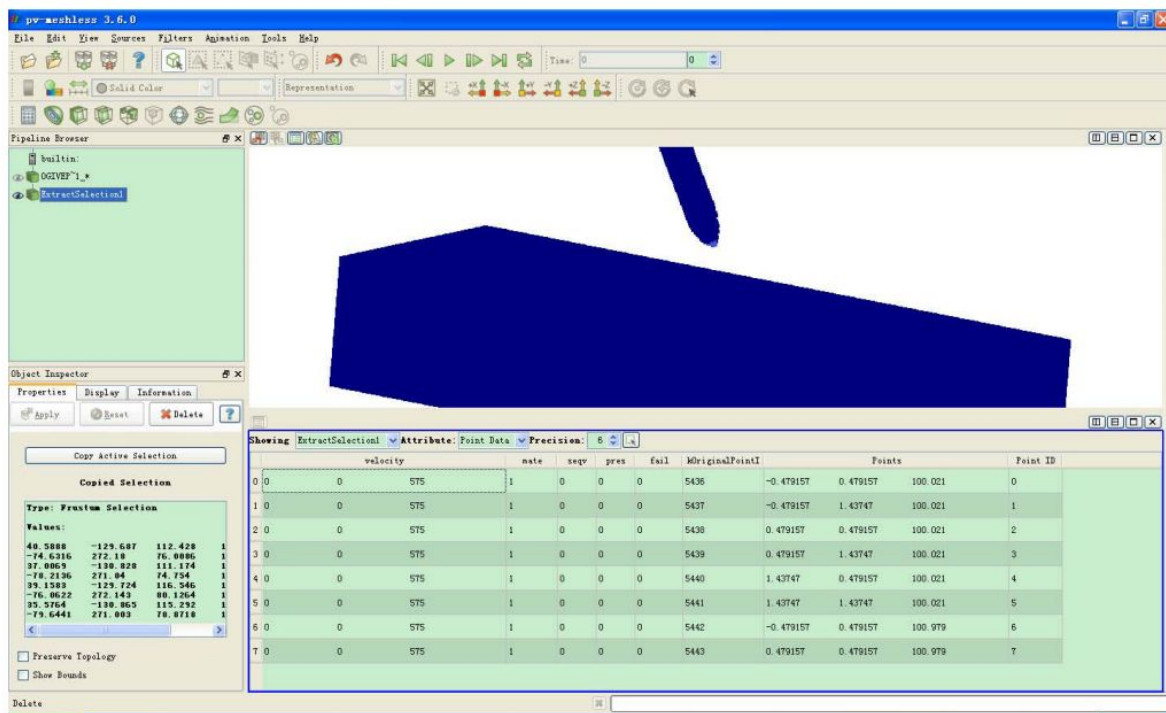5. Slice, cut, view internal points.

图 5.36: extract