

TenGun™

User Guide

Version 2.5

Last Updated October 8, 2023

TRADEMARKS

TenGun is trademark of Nippon CompuTools Ltd.

Windows is a registered trademark of Microsoft Corporation.

Other products mentioned may be trademarks or registered trademarks of their respective owners.

COPYRIGHT

Copyright © 2018-2023, Nippon CompuTools Limited. All rights reserved. This document contains confidential information and trade secrets of Nippon CompuTools Ltd. Use of this document is governed by written agreement with Nippon CompuTools Ltd. While this work is copyrighted, this does not imply publication or disclosure of the work. No part of this document may be reproduced, transcribed, stored in a retrieval system, translated into any language, or transmitted in any form or by any means, electronic, mechanical, photocopying, recording, or otherwise, without the prior written permission of Nippon CompuTools Ltd.

First printed August 29, 2018.

Portions of TenGun are based on public domain software. The relevant licenses are documented in the file ThirdPartyLicenses.pdf, which is installed with TenGun.

NOTICE

Nippon CompuTools Ltd. makes no representations of any kind whatsoever with respect to the contents hereof and specifically disclaims any implied warranties, merchantability, or fitness for any particular purpose. Nippon CompuTools Ltd. shall not be liable for errors contained herein or for incidental or consequential damage in connection with the furnishing, performance, or use of this document or its contents.

The software described in this document is furnished under a license and may only be used or copied in accordance with the terms of such license.

Nippon CompuTools Ltd. reserves the right to revise this document from time to time and to make changes in the content hereof without obligation to notify any person of such revisions or changes.



Table of Contents

Overview	1
Getting Started	2
License Installation	2
Command Line Options	3
Input File Formats	4
Output File Formats.....	6
CAD Files.....	6
Analysis Application Data Files.....	6
Point Data Export.....	7
Screen Layout & Navigation Tricks	8
Time Series Mode (Post-Processing).....	11
User Interface.....	12
File Open	12
Automatic Profile Selection	13
Fields	14
Pan, Rotate, and Spin.....	14
Zoom	15
Camera Position	15
View Features and View Type	16
Vector View	16
Choose the vector to use.....	16
Filters.....	18
2D Cut.....	19
3D Cut.....	20
Screenshots.....	22
Animation & Movies	22
Charts	24
Reports	26
Report Setup Example	26
Object Picking.....	27
Hide Object.....	27
Pick Interior Face.....	28

Distance Measurement	28
Profiles	29
General Settings	29
Continuous Scalar Colors Settings.....	31
Scalar Bar Label Format	32
Discrete Colors Settings.....	33
Field Settings	34
Chart Settings	37
Animation Settings	38
Help	40
Additional Documentation	40
Assembly Mode (Pre-Processing)	41
Generating Data	42
Revolved and Extruded Surfaces.....	44
Part Colors	46
Exporting Data	47
Export Model as Point Cloud.....	47
Export Analysis Data	50
Assembly Example – Part 1: Assemble the Parts.....	53
Assembly Example – Part 2: Create Analysis Data	58
Object Editing Tools	59
Analyze Mesh	59
Analyze Point Density.....	59
Background Mesh.....	59
Points to Surface	60
Remove Outliers	61
Subdivide Triangles	62
Object Alignment	62
Join Faces Operation.....	63
Boolean Operations	64
Advanced Cut Operations.....	65
Create Hollow Solid	66
Repair Shape.....	66
Subdivide Faces	66
Subdivide Surfaces.....	66

Edit Geometry	67
Image Display	69
Text Editor	70
Performance Hints.....	71
Experimental Features	71

Notation Used in this Document

- Key sequences that you may type are enclosed in a box. For example, indicates that you should press the “Enter” key.
- Dialog buttons are enclosed in a box with a gray background. For example, where this document says “Select ”, you should click the OK button or navigate to the OK button and press the key.
- Menu sequences are indicated by one or more items with a gray background (and no box), and separated by an arrow character “→”. For example, to open a file you navigate with the mouse or the keyboard to the → dialog.
- Sections that apply only to Time Series Mode are marked with the  icon, and those that apply only to Assembly Mode are marked with the  icon. These modes are explained later.

Overview

TenGun is a high-performance, easy-to-use tool for working with Point Cloud data, and is available for Windows, Linux, and macOS. It is primarily targeted for use with SPH (Smoothed Particle Hydrodynamics), CFD (Computational Fluid Dynamics), and similar analysis applications.

TenGun (点群, pronounced like "Ten Goon"), literally means "Point Cloud" in Japanese.

This is not to be confused with the mythical Tengu (天狗) creature, which is what everyone insisted on hearing when I suggested this name.

- The author -

TenGun has the following usage scenarios:

- Post-Processing, which we call "Time Series Mode". This mode is used to graphically inspect the output of an analysis application.
- Pre-Processing, which we call "Assembly Mode". This mode is used to assemble the parts from a CAD application and to convert them to the input format of the analysis application.
- Text Editing. The editor can work with files either related to your analysis work as well as files some other, unrelated use.

These usage scenarios are discussed in the following chapters.

Getting Started

To start, follow these steps:

- Install the program by opening TenGunSetup.msi. This is a straightforward Windows setup application.
- Obtain and install a License (refer to [License Installation](#)).
- Run TenGun, either from the Desktop shortcut, the Start Menu shortcut, or from the command line.
- If you are using *simple data files that do not contain a field information header*, you must register the list of fields in your data. This can be done in the [Edit → Settings](#) screen.

License Installation

TenGun is available with the following licensing options:

- Free. Includes the basic Time Series Mode viewer functionality and supports a limited number of particles. It may be used for non-commercial use, and no support is provided.
- Advanced Viewer. Time Series Mode viewer, no particle limit, may be used for commercial use, and support is available.
- Reporter. Same as above, and allows creation of Reports and Charts.
- Professional. All features including Assembly Mode pre-processing features.

The "Free" version does not require a license file.

Advanced Viewers features and above require a valid software license to operate. To set up a license:

- Start TenGun and go to the [Help → About](#) window.
- Click the "Request License for MAC Address xx-xx-xx-xx-xx-xx" link. This will send an email to us to request a license.
- You will receive a reply with an attached file named "TenGun.lic". You may manually copy this to "C:\ProgramData\TenGun", or you may do the following:
 - Drag & Drop TenGun.lic to any folder, for example your Desktop folder.
 - Start TenGun if it is not already running.
 - Drag the license file into the TenGun main window.
 - TenGun will install the license file for you.

NOTE: You cannot directly Drag & Drop an email attachment to install it. However, you may first move the attachment to the file system (e.g., to the Desktop), and *then* drag it into TenGun.

Command Line Options

The only command line option is currently “-lang”. The default language is the same as the System language. To override the default specify “-lang en” for English, or “-lang ja” for Japanese.

Input File Formats

TenGun supports the following types of input files for Time Series and Assembly Modes.

Type	Extension	Description	Use Case
Plain data	.dat	Simple data file with no information about the structure.	Time Series Mode from analysis output.
BMP	.bmp	Windows BitMaP image file.	
CSV	.csv	"Comma Separated Values". The first line of the file is expected to contain the names of the fields, and must minimally include fields X, Y, and Z. CSV files can be imported by many applications including spreadsheets.	Time Series Mode from analysis output.
DICOM	.dcm	Digital Imaging and Communications in Medicine (Xrays, CT-Scans, MRI scans).	
IGES	.igs, .iges	"Initial Graphics Exchange Specification". A standardized format for exchanging CAD drawings.	Assembly Mode from CAD exchange.
JPEG	.jpg, .jpeg	Joint Photographic Experts Group image file.	
LS-DYNA		LS-DYNA d3plot. There are a huge number of variations for this format, so this is currently an Experimental Feature .	
OBJ	.obj	A geometry definition file format originally developed by Wavefront Technologies that contains geometry and texture information.	Assembly Mode from CAD exchange.
PLY	.ply	"Polygon File Format" or "Stanford Triangle Format". Simple format originally designed to store 3D-Scanner data. Binary and ASCII formats.	Assembly Mode from CAD exchange.
PNG	.png	Portable Network Graphics image file.	
STEP	.stp, .step	A widely used CAD exchange format standardized under ISO 10303. TenGun supports the ISO 10303-203 and ISO 10303-214 variants, which are used in mechanical and automotive design, respectively. Some other variants may work.	Assembly Mode CAD exchange.
STL	.stl	A " <u>S</u> tereol <u>i</u> thography" file. Contains simple 3D triangulated surfaces but not much other information. Binary and ASCII formats.	Assembly Mode from CAD exchange.
VTK	.vtk	A legacy "Visual ToolKit" file. Binary and ASCII formats.	Data exchange with other

			graphics applications based on "VTK".
VTP	.vtp	A "Visual ToolKit" XML-based file. Binary and ASCII formats.	Data exchange with other graphics applications based on "VTK".

Hints

- If you are working with CAD data in TenGun, we strongly recommend that you use STEP format if possible. Although STL files are common, it is not a good format for CAD models. STL was designed to store 3D-Printer data, and these files do not contain design attributes that we need to perform advanced editing operations. Furthermore, STL handling can be slow because even seemingly simple objects can be represented internally by many thousands of triangles.
- LS-DYNA "d3plot" files is just one of the files created by an LS-DYNA analysis. For TenGun to read LS-DYNA data you must provide *all* of the LS-DYNA output files to TenGun, in addition to "d3plot".

Output File Formats

CAD Files

TenGun can write IGES, STEP, STL, VTK, VTP, and X3D or X3DB file formats. This is useful for two purposes:

- Save your Assembly as you build it. If you need modify your data later, the saved assembly is your backup.
- Export your Assembly in a different format for use with another application.



Menu: File → Save Assembly.

STEP and IGES files can contain multiple parts.

As for STL:

- STL files can only contain a single part. Therefore, if you try to save a multi-part assembly in STL format, TenGun will create multiple files with “-1”, “-2”, etc. appended to the output file name that you specify.
- If you export a STEP or IGES file as STL, the output will consist of a multitude of triangles. You cannot import an STL file and expect to then convert it back to the same STEP or IGES object because original connectivity information is lost in the export process.

STL, VTK, and VTP files are by default written in binary format. Select the **Save as Text** on the “Save Assembly As” dialog to write the files as ASCII Text.

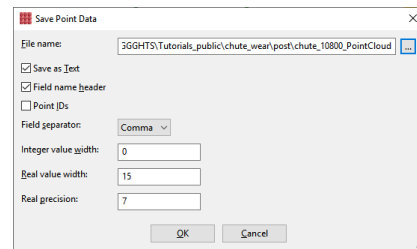
Finally, TenGun can export an assembly as an X3D (.x3d) or X3D Binary (.x3db) file. X3D is the successor to VRML. At this time we cannot read X3D files, so we do not claim that we can convert between this and other formats.

Analysis Application Data Files

TenGun can convert your assembly to a data file suitable as input to an analysis application. Requirements for this are specific to that analysis application and are documented separately.

Point Data Export

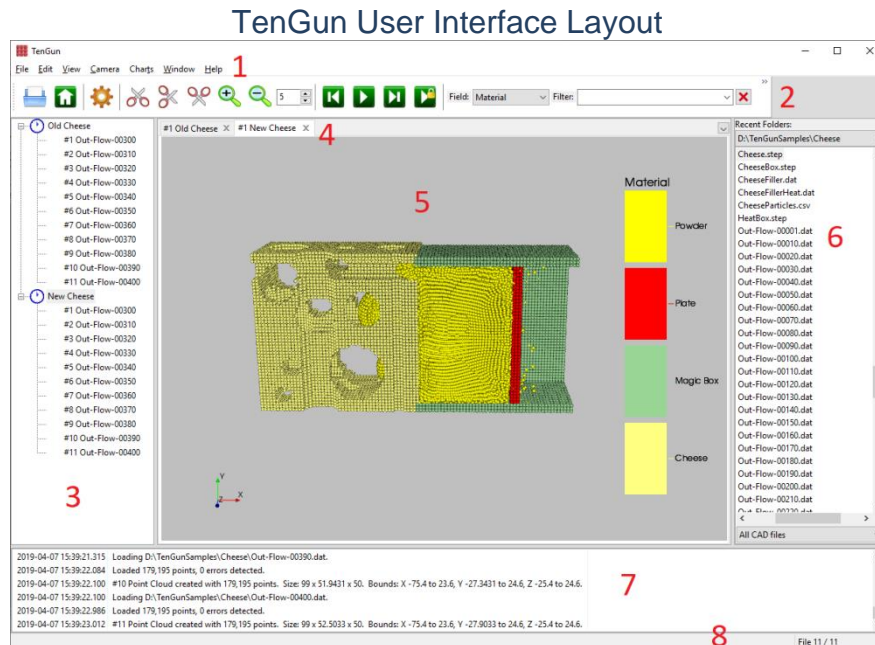
🕒 Time Series point data can be saved in a text file using **File** → **Export** → **Selected Point Data**. This function saves the currently displayed (i.e., filtered) data, allowing you to export a subset of a Time Series for use in another application. The default settings will create a CSV file that can be immediately loaded into a spreadsheet.



You may save the data as a CSV (default), DAT, OBJ, PLY, STL, VTK, or VTP file. By default, the output is written as ASCII text. You can also save PLY, STL, VTK, and VTP files in binary format by unchecking the **Save as Text** option.

Screen Layout & Navigation Tricks

Below is a screenshot of TenGun in action.



Key elements of the User Interface (numbered in red) are:

1. Menu Bar. Sub-menus are:

- File. File-related operations and Exit. There are additional items when in Assembly Mode.
- Edit. Some editing operations.
- Camera. Change the viewpoint.
- Object. Create CAD objects and modify object properties in Assembly Mode.
- Tools. CAD object and Point Cloud utility functions, for example, to edit geometry.
- Charts. Create charts.
- Window. Change window settings.
- Help. Display help documentation and program information.

2. Toolbar. Contains buttons you can click to perform some common operations instead of using the menus.

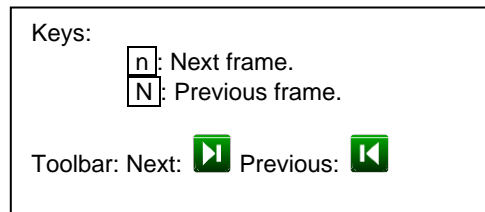
3. Tree View. There is one root entry for each Panel Tab. You may edit the name corresponding to a Panel Tab; this will also update the name on the Panel Tab.


4. Panel Tabs. One tab per set of files. There are two tabs in this example.

5. Scene Panel(s). Each panel represents the graphics window for one set of files, with two panels shown here. See [Hints](#) at the end of this section for how to display panes side-by-side.
6. File Browser ([Experimental Feature](#)). This control lets you browse through the file system for your data. There are three parts:
 - At the top is a dropdown list of the folders you have most recently accessed from TenGun. Using this list is often quicker than navigating the folder tree. The folder you select may sometimes appear near the *bottom* of the tree view.
 - The middle portion shows tree-structured view of the file system that you can navigate using either the mouse or the keyboard.
 - At the bottom is a list of file extension filters that you can use to limit the output.
7. Message Window. TenGun logs informational messages in this area.
8. Status Bar. Shows some User Interface status messages. Of particular interest is the bottom right corner which displays “File n / m”, indicating the progress of file reading operations.

Hints

- You can change the width of the File Browser by dragging the vertical bar sideways. You can also entirely hide this from the **Window** → **Show File Browser** menu item.
- When you have multiple files and multiple panels open you can navigate as follows:
 - To change panels, left-click a Panel Tab, or click on the corresponding item in the Tree View.
 - To jump directly to a specific Time Series item, left-click on the item in the Tree View.
 - To go to the next item in a Time Series view press **[n]** ("Next"), and to go to the previous item press **[N]** ("Shift Next").
- For a Time Series, Panel Tab title includes “#1”, “#2”, etc. as a hint to which file in the series you are currently viewing.
- A different icon is displayed in the Tree View for Time Series data (🕒) and Assembly Mode data (⚙️).
- To combine scene panels for a side-by-side view, drag the tab for one panel to the left, right, or bottom edge of the target pane. The target pane will be split so that you can see multiple models in a single panel.



- To split scene panels, drag the tab for one panel into the Tab Bar. The combined model views will be split into individual panels.
- With scenes in a side-by-side view, you can scroll the scenes in parallel by clicking the  ("Synchronize Scenes") button on the toolbar. This is very handy for seeing how different data fields are related. Click the button again to disable parallel scrolling. When parallel scrolling is enabled the icon changes from green to red.
- Sometimes you want to compare two sets of results, so you open lots of files with similar names, but from different folders. You can get totally confused, just by the names, and you can make it worse if you drag two tabs so that they appear side-by-side in the same window.

To avoid such confusion, first load one set of files. Next, left-click the scene's root item in the Tree View. Wait a second, and click again so that the label will enter edit mode. Change the label to something better than the file name such as "Old". Changing the Tree View label will also change the tab label. Now repeat this for the other set of files, but use a different label such as "New". Confusion over the file names is now gone even if you rearrange the tabs.

Time Series Mode (Post-Processing)

Analysis applications typically generate a number of time step-based particle (point) output files. The output contains particle positions, velocities, pressure, temperature, and other relevant analysis results. TenGun is then used to visually verify those results.

Post-processing features include:

- Flexible input file handling.
- Filter values using field name and range expressions to display specific details.
- Cut (slice) the results one or more times so that you may inspect the inside of the model.
- Emphasis on ease-of-use, and without a bloated User Interface.
- Easily create analysis presentation material.
- Animated playback. Play back a series of analysis output files.
- Create a movie file from the animation output.
- Automated report creation.
- Multiple chart types: Frequency Distribution (bar chart or line chart), Line Chart, Scatter Diagram, and Timeline with statistics.
- Save Screenshot to file.
- Copy Screenshot to Clipboard for immediate use in another application.
- Easily customizable "Display Profiles" that allow definition of user-specific input file layouts, colors, labels, and other settings.
- High-performance, even with large datasets.
- Multi-lingual. English and Japanese are currently available.

To use Time Series mode, we suggest that you next review the User Interface chapter which follows.

User Interface

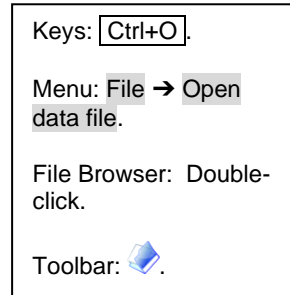
This section describes key features of the user interface.

File Open

To open a file in the Text Editor, navigate to the **File** → **Open text file** dialog.

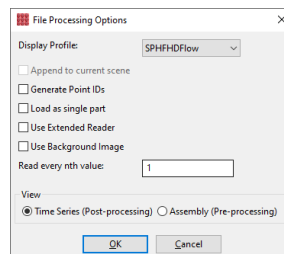
To open a file for Time Series or Assembly mode display do one of the following:

- Double-click a file name in the built-in File Browser, or
- Select multiple files in the built-in File Browser and press **Enter**.
- Drag and Drop a file from another application such as Windows Explorer, or
- Navigate to the **File** → **Open data file** dialog. Select the file(s) and select **OK**.



To select multiple adjacent files, hold the **Shift** key while picking them; to select multiple non-adjacent files hold the **Ctrl** key while picking them.

Next, TenGun will display the File Processing Options dialog:



The File Processing Options are:

- Display Profile. If you are just starting, try the "Default" profile. Later, you will learn how to customize the display parameters. Display Profile settings apply mainly for Analysis Output files, and are especially important if processing "dumb" input files containing no field information. For CAD files the profile mainly only affects the coloring, so you may choose any profile. See also [Automatic Profile Selection](#) below.
- Append to current scene panel. Check this to add the file(s) to an existing panel; otherwise a new scene panel will be created.

If a CSV or DAT file does not display, you may need to customize the [Field](#) definitions in the profile.

- Generate IDs. Check this to enable the automatic creation of Point ID data. When your analysis is misbehaving, it is sometimes helpful to filter by the ID so you can display the properties of particular particles.
- Load as separate parts. If this option is checked, CAD files loaded in Assembly Mode will be treated as separate parts, and each part can be modified and manipulated with TenGun's geometry-related tools. If unchecked, multiple parts are combined into a single object.
- Use Extended Reader. This option currently applies only to OBJ and STL files. The default readers are fast. However, to use OBJ or STL data with Boolean Operations and other advanced Assembly Mode features you must use the slower "extended" readers.
- Use Background Image. Check this to use the background image as set in the profile. This is disabled by default because under normal usage it is rather obtrusive. However, you may want to use a background image when creating presentation materials.
- Read every nth value. This option skips some of the input when processing text input files. A value of "1" reads all input, but a value such as only "5" reads only every 5th value. This is a great speedup, and can save a lot of memory if you working with huge data files.
- View. Select the display mode: Time Series, Assembly, or Text Edit as appropriate. The available operations depend on this mode, so remember to select the correct mode.
- After selecting , TenGun will try to read and display the data.

Automatic Profile Selection

If you work with many different output file formats, you can set TenGun to automatically select the correct profile based on a file name pattern. To use this feature:

- Create a text file name C:\ProgramData\TenGun\ProfileMap.dat.
- Add one line for each file pattern that you use. Each line should contain a *regular expression* (a wildcard pattern) to match the name, a space, and the name of the profile. For example:

Out-Flow-.....\.\dat Flow

The first five periods match any character. The "\." is an *escaped* period, so instead of matching anything, the next character in the file name must be a real period. When you try to open a file matching this pattern, TenGun will automatically select the profile named "Flow" in the File Processing Options dialog.

Blank lines and lines beginning with "#" in ProfileMap.dat are ignored. Also, this feature does not currently support international characters in the name pattern

or the profile name.

Notes

- The number of files you can open is limited only by the available memory.
- When TenGun displays Time Series data, the camera is positioned such that the entire scene is visible, and the focal point is the center of the scene. This works fine if the bounds of the scene are the same in each time step. However, some data is unconstrained (especially Discrete Element Method output), and the scene boundaries are different at each time step. In this case, the camera would need to be moved at each time step, but that would result in rather strange animations. TenGun solves this by computing the overall bounds of all scenes. When the last file of a series is loaded, TenGun may redraw the currently visible scene to adjust the camera position, so don't be surprised if the current scene suddenly appears to shrink.

Fields

🕒 To view details for particle data, select the field name in the "Field" dropdown list. TenGun will then display a Scalar Bar. For scalar values you can then see the range of values (e.g., for Density or Pressure), and for discrete variables you can see the corresponding color legend (e.g., which color represents fluid or various solid objects).

Field Handling Rules

- Field name matching is case-insensitive. Field names may contain alphanumeric characters and “_”. However, you should avoid using “_” at the beginning of a name to prevent possible conflicts with TenGun internal data.
- Extra fields in a Profile that are not used in a particular input file are ignored. You may want to keep extra field definitions in case you work with a variety of similar data.
- If a CSV input file contains an undefined field, TenGun will automatically add it (temporarily) to the Profile as a *float* vector. If this is an inappropriate type, please add an appropriate field definition.
- X, Y, and Z (particle positions) are **required** input fields. These are implicitly defined in the Profile with a Data Type of "*float*".
- Maximum line size for text input is 4096 bytes; lines are typically 200 bytes long or less.



Pan, Rotate, and Spin



You can pan, rotate, and spin the model using the mouse. Mouse button usage depends on the type of mouse, but the Left Button is used for rotation in all cases.

You can access a Quick Reference guide in the TenGun Help menu that describes mouse operation as well as summarizing keyboard actions.

Zoom

You can zoom in or out using either keys or the Toolbar buttons.


Keys: Zoom In: , Zoom Out: .












Toolbar: Zoom in: , Zoom out: .

Camera Position

Use the keys and the Camera Menu to quickly move camera around.

The Rotate actions “x” and “y” rotate the camera 10 degrees counterclockwise about the respective axis, while “X” and “Y” rotate clockwise.

The  (“Home”) button on the Toolbar can also be used to undo all edits, reset the zoom level, and reset the camera to the front view.

Keys: Top: , Left: , Right: ,
, Front: , Rear: ,
Bottom: , Rotate +X: ,
Rotate -X: , Rotate +Y: , Rotate -Y: .

Menu: Camera → Front, Left, Right, Top, Bottom, or Back.

Toolbar: .

View Features and View Type

Use the View menu to display a model view mode:

- Points. Display the model as points (pixels).
- Spheres. Display the model as spheres. Spheres in general look cuter than points, but require more processing power to draw.
- Solid. Display a CAD model as a solid object.
- Vectors. See discussion below.
- Wireframe. Display a CAD model as a wireframe.
- Bounding Box. Add a bounding box around the data. This is sometimes helpful in visualizing particle flow if the data is not constrained by a container.
- Lighting. Toggle lighting effects.
- Scalar Bar. Enable or disable display of the Scalar Bar.

Keys:

v	b	: View Bounding Box.
v	h	: View Spheres.
v	p	: View Points.
v	s	: View Solid.
v	v	: View Vectors.
v	w	: View Wireframe.
s	s	: Toggle Scalar Bar display.

Menu: View → Bounding Box, Points, Spheres, Solid, Vectors, or Wireframe. Also, Lighting, Scalar Bar.

You can use the Point Size control on the Toolbar to change Point and Sphere sizes. Make the points larger to fill in gaps, or make them smaller to peek inside the model.

Vector View

 This view displays a field as a scaled vector instead of a point or sphere.

Individual fields may be combined into 2D or 3D vectors. For example, the three fields VX, VY, and VZ can be combined to create the 3D velocity vector. These vectors are defined in the General Profile settings. Individual components of the vector are separated by commas (e.g., “VX, VY, VZ”), and multiple vectors are separated by semicolon (e.g., “VX, VY, VZ; StressX, StressY, StressZ”).

Usage is then as follows:

- Select the View → Vectors menu item or press .

Choose the vector to use.

- Set a scale value.
- Select .

TenGun then normalizes the selected vector magnitudes to the range 0 to 1. For each field value, a line is drawn from the field’s position in the direction of

vector. The length of the vector is controlled by the “scale” value in the settings dialog. Note that the scaled vector length is in “World Coordinates”, so you may need to make the scale value small if the particles are small.

In general, you use coloring to highlight features of your data such as temperature or density. The Vector View feature can sometimes be combined to make interesting graphics that show not only something like temperature, but the particle flow.

Filters

🕒 Filters allow you to display a subset of the input data. To apply a filter, enter a *Filter Expression* in the Filter area on the Toolbar and press **Enter**. TenGun remembers your most recent filters, which you may select from the dropdown list instead of typing the expression again.

A *Filter Expression* is of the form:

field relop constant

where:

- *field* is a Field Name. Field names are case insensitive.
- *relop* is a relational operator: < (less than) <= (less than or equal), = (equal), != (not equal), >= (greater than or equal), and > (greater than).
- *constant* is a constant numeric value.

Examples:

Pressure > 1.5

Material = 3

You may use multiple expressions by separating them with the logical operators "**and**" and "**or**". For example:

material != 1 and density > 0.95

Expressions are evaluated left-to-right. You may currently not group sub-expressions using parentheses.


2D Cut

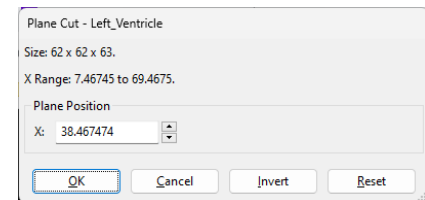
🕒 You can look inside a model using the 2D (Plane) Cut feature:

- Select one of the Cut directions as shown in the sidebar.
- A yellow plane is displayed perpendicular to the cutting direction. There are two ways to move the cutting plane:
 - Left-click and drag on the plane. This method is easy and helpful when you just need a rough cut. You may need to rotate the model slightly in order to grab the plane.
 - Or, directly set the boundary values in the “Plane Cut” dialog. This method can be used when you need a more precise cut.
- A normal vector is also drawn through the plane pointing in the direction of the portion to be *preserved* after the cut.
- You can rotate the cutting plane by dragging one of the cones on the ends of the normal vector.
- Press **Enter** to complete the cut, or press **Esc** to cancel.
- Undo a cut operation by typing **Ctrl+Z** or from the **Edit** → **Undo** menu item.

Keys:

C	X	: Cut left.
c	X	: Cut right.
C	Y	: Cut below
c	Y	: Cut above
C	Z	: Cut front.
c	Z	: Cut rear.
i		: Invert cut direction.

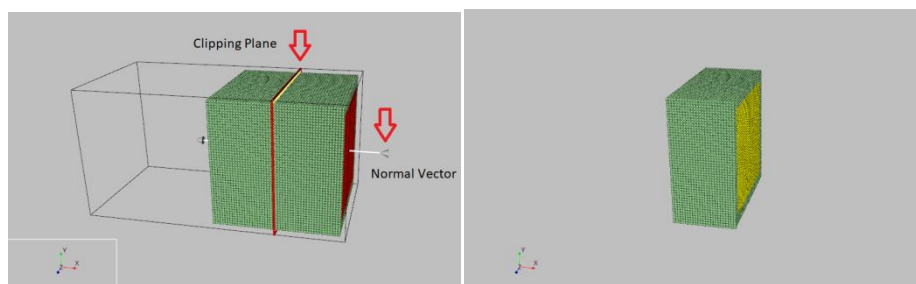
Toolbar:  for cx, cy, and cz. Press Shift while clicking to use cX, cY, or cZ.



The 2D Cut feature is currently not available in Assembly Mode.

2D Cut Example

The image on the left shows the screen after typing the **C** **X** (“Cut right”) command. The cut will be made at the Clipping Plane, and the portion in the direction of the Normal Vector is preserved. The image on the right shows the result.



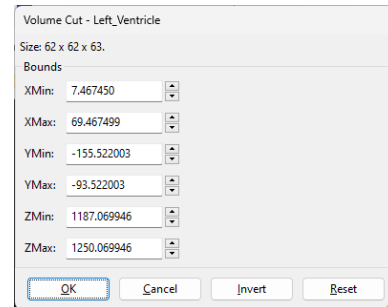
3D Cut

🕒 The 3D Cut feature allows you to cut out portions of an object so that you may view the inside. To use this feature:

Keys:

C **V**: Cut Volume.
I: Invert cut direction
(toggle between inside and outside).

- If you are in Assembly Mode, select the object of interest by Right-clicking it.
- Type **C** **V** (“**C**ut **V**olume”).
- A bounding box is displayed around the object and the “Volume Cut” dialog is displayed.
- There are two ways to select the desired volume:
 - Use the six round “handles” on the outsides of the bounding box. Left-click and drag any of the handles to move the associated face in or out. This method easy and helpful when you just need a rough cut.
 - Or, directly set the boundary values in the Volume Cut dialog. This method can be used when you need a more precise cut.
- Type **I** (“**I**nvert”) to toggle the selection direction. This allows you to keep the outside of the bounding box instead of the inside.
- When you have enclosed the area of interest, press **Enter** or click the OK button. To cancel the operation press **Esc** or click the Cancel button.
- Undo a cut operation by typing **Ctrl+Z** or from the **Edit** → **Undo** menu item.

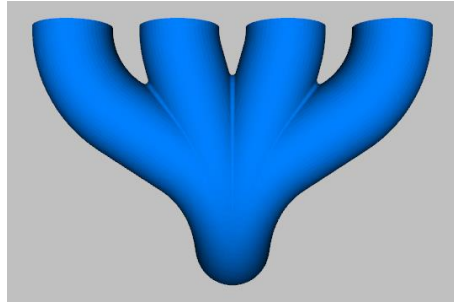
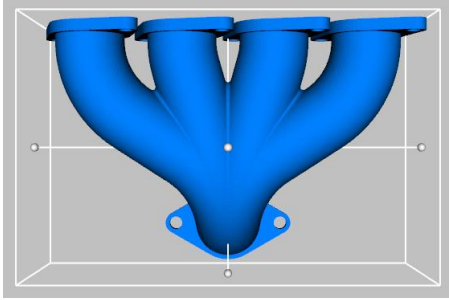


Hints

- 3D Cut is especially useful when creating Point Cloud. Some portions of the model can often be removed to reduce the number of particles in the analysis without having any effect on that analysis.
- In Time Series Mode, a 2D-Cut is applied to all members of the Time Series. At this time, a 3D-Cut in Time Series Mode is applied only to the current Time Series item. This behavior is likely to change in a future version so that the cut is applied to all Time Series items.

3D Cut Example

The left image shows an exhaust manifold with 3D-Cut active. The top and the rear handles are moved down and in to remove the flanges. The right image shows the result after cutting.



Screenshots


You can make screenshots in the following ways:


- Press **Ctrl+S**. You will be prompted for the name of a file to save the screenshot in. You may also select the file format (BMP, JPEG, PNG).
- Press **Ctrl+C**. The screen image is placed in the Clipboard, and from there you may paste the image into some other documentation.

Hint: Screenshots taken on a wide screen will generally have a lot of empty space. We suggest that you resize the TenGun window so that the scene area has an aspect ratio of approximately 4:3 (1.333). This aspect ratio is displayed in the Message Window as you resize.

This advice also applies to creating reports and animation. Use the Window → Reset Window Layout menu item to restore the size to the default.

Animation & Movies

 To create an animation:

- Open two or more files. Analysis programs typically create the output files with a sequence number such as output001.csv, output002.csv, etc.
- The first file will be loaded immediately, and additional files will be loaded in the background. NOTE: You can see the file loading progress by observing the Status Bar. The bottom right corner of the screen shows a load progress message, such as "File 15 / 20", which indicates that 15 out of 20 request files have been loaded.
- Optionally set a custom title message. See [Animation Settings](#).
- Type **a** or click the  ("Play") button on the Toolbar. This functionality is disabled until at least two files have been loaded.
- The first time a file is displayed, TenGun will display the underlying graphic data, and it will also create an image file of that data. If you continue playing the animation, the image file is displayed instead of the graphic data, which is considerably faster. The first pass is therefore likely to be relatively slow, but subsequent passes are more likely to match the per-scene Animation Time that you have specified.
- The Animation playback time can be set in the [Animation Settings](#). The default time is 200 milliseconds between screens.
- To stop playback press **Esc**, or press the Stop Animation button in the Toolbar.

Keys: **a** to start, **Esc** to stop.

Toolbar: Start:  Stop: .

Image files are stored in the "TenGun_Animation_*datetime*" subfolder where the input data resides. This folder will be deleted whenever you open a new input file or when you exit TenGun. If you want to keep the bitmaps, copy them before you do one of these actions.

To create a movie:

- First create the animation data as described above.
- Select the **File** → **Save Animation as Movie** menu item.
- Enter the movie file name. The file type can be “avi”, “mp4”, or “ogg” as indicated by the file extension. The “mp4” and “ogg” formats are supported in a web page in a <video> control. The “ogg” format tends to be the smallest but is not supported by Safari as of this writing. If you set the type to “all”, TenGun will create movies in all supported formats.
- Select OK to create the movie.

Notes

- When animation starts, the Scalar Bar range for continuous variables is calculated from all of the data that has been loaded *so far*. This is done so that the Scalar Bar shows the variable range for the entire simulation rather than the range for individual scenes. If the exact range is important to you, you must wait until *all* files have been loaded.
- The default “on” behavior is to use the overall minimum and maximum values from all scenes in the Scalar Bar. The command can be used to toggle this behavior. When “off”, the minimum and maximum of individual scenes is used instead.
- While animation is active you may not perform any actions that affect scene layout such as zoom, filter, and so on. Therefore, you should set up the display exactly as you want it *before* you start the animation.
- TenGun creates movies by running the command file “C:\ProgramData\TenGun\CreateMovie.cmd”. This command file is responsible for running a separately installed third-party application such as the open source [FFmpeg](#). This application then converts the animations images created by TenGun into the desired movie file format. Feel free to customize this command file to meet your requirements.

Some media players including Windows Media Player or VLC automatically play a movie when it is opened, and often do not display the very beginning. If this occurs, rewind the movie and replay from the beginning. This problem does not occur with the ffplay player included with FFmpeg.

Charts

 You can create these types of charts:

- Distribution Chart. This calculates a Frequency Distribution and displays it either as Bar Chart or a Line Chart. Multiple variables may be selected.
- Line Chart. You may plot a single variable along X-Axis to plot against one or more variables on the Y-Axis. Either axis may be plotted using a logarithmic scale instead of the default linear scale, although the internal charting code may sometimes override your choice.
- Particle History. This plots time-series values for one or more variables and one or more particles. For points you may specify:
 - A single point. Point IDs start at 1.
 - A point range where the values separated by a hyphen (e.g. 2000-2005).
 - Multiple points or point ranges separated by a comma (e.g. 100,200-204,300).

The point set is initialized to the most recent five points that you have selected with the [Point Picker](#).

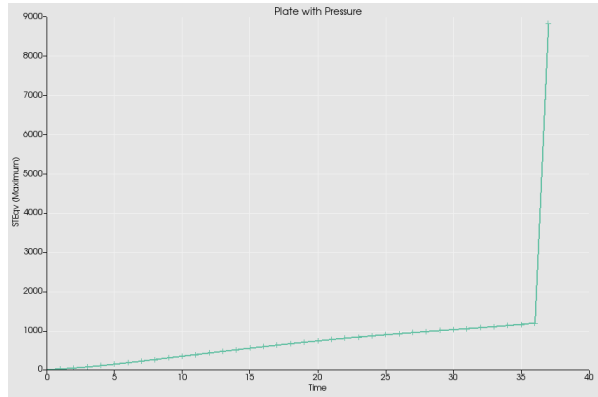
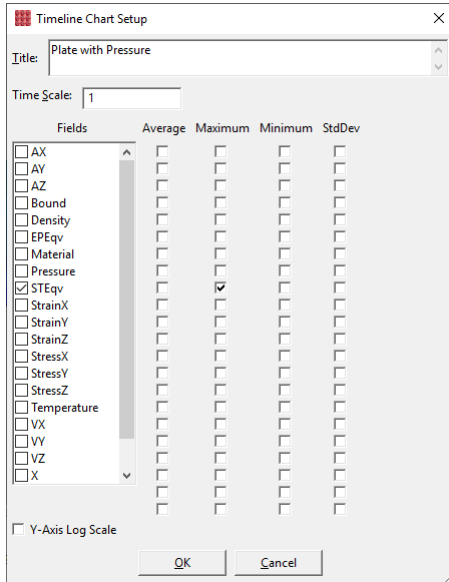
- Scatter Diagram. Same options as for Line Chart, but X-Y pairs are presented as dots instead of lines.
- Timeline Chart. This plots a time-series for one or more variables for the selected statistic(s). Available statistics are average, maximum, minimum, and standard deviation. This chart is useful for fields such as Equivalent Stress where only the maximum value is actually useful.

To create a chart:

- Open a Time Series view. Particle History and Timeline Charts require that you open multiple files.
- Select a chart type from the Charts menu. The data source is in the Time Series data, so you cannot select a chart type from an Assembly Mode or from a Chart view.
- Fill in the parameters appropriate to the chart. Minimally this involves selecting the field(s) to be displayed in the chart.
- Select . The chart is displayed in a new tabbed pane.

Timeline Chart Example

The following shows the setup dialog for a Timeline Chart for a stress analysis and the resulting chart.



Reports

🕒 TenGun can create a report, with graphics, of your data. To do so:

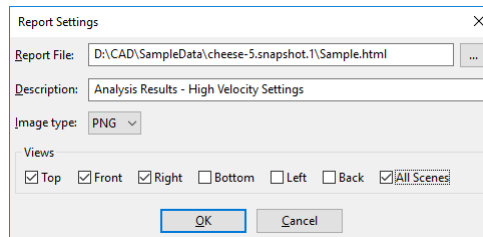
- Open a file or a file series.
- Optionally set a Filter.
- Select the **File** → **Create Report** menu item.
- Enter a report file name and optionally a description. The description is a good place to enter notes about conditions used for this analysis.
- Select the file type for image files (BMP, JPEG, or PNG).
- Select the fields you wish to report on.
- Select one or more object faces to be used to create graphics (Front, Back, Left, Right, Top, Bottom).
- Check "All Scenes" to include all scenes of the time series in the report. If unchecked only the current scene is included.
- Select **OK**.

TenGun then moves the camera as required, cuts the model in half, and takes a screenshot for each selected face.

The generated report file can be viewed by any HTML5-compatible browser.

Report Setup Example

The Report Settings dialog is shown below.



Object Picking

TenGun provides the following object picking behaviors:

- **Points.** The Point Picker displays all properties for the *nearest* selected point in popup "balloon". (To hide the balloon, move the mouse pointer.) Use the Zoom, Rotate, and Cut functions as necessary if you actually need to pick a specific point.
- **Edges.** Pick edges of a CAD model.
- **Faces.** Pick faces of a CAD model.
- **Vertices.** Pick vertices of a CAD model.
- **Object.** Right-click an object in Assembly Mode to select it, and the object is then highlighted. You may use the [Geometry Editor](#) to change position, scale, and rotation of individual objects. And, you may combine multiple objects using [Boolean Operations](#). You may also select objects via the Tree View, which is sometimes helpful when one object obscures another object.

Keys: **p** **p** (pick points), **p** **e** (pick edges), **p** **f** (pick faces), or **p** **v** (pick vertices) to enable picking mode, **Esc** to turn off picking.

Mouse: Left-click to pick a point, edge, or face. Right-click to pick an object. To select multiple items hold the **Shift** key

To select multiple objects, press **Shift** while clicking. To deselect an object, click it again.

Hints

- To rotate an object while picking is enabled, move the mouse pointer outside of the object to a blank area. While pressing the left mouse button, move the mouse point to rotate the object.

Hide Object

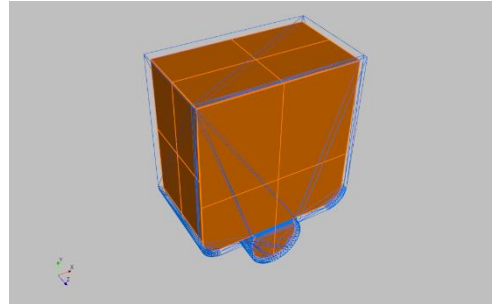
You may sometimes wish to hide an object, especially when you want to edit one object contained inside another object. To do so:

- Navigate to the object that is in the way type typing **n** ("Next"), **N** ("Shift Next" for previous), or by clicking the object in the Tree View.
- Type **h** ("Hide").

To restore visibility, navigate to the hidden object and type **h** again. You can also unhide the object by clicking it in the Tree View.

Pick Interior Face

Sometimes you need to select an interior face of a CAD object so that you can set properties such as a boundary condition. To do this, first enter Pick Face mode as previously described. Next, you normally click an exterior face. However, to select the “nth” face behind that exterior face, type a number (1 to 9) before clicking the exterior face. The most common case is that you type “1” to select the inside of a wall, and from there you can set properties for that inside wall using Edit Part Properties (refer to [Assembly Mode \(Pre-Processing\)](#) for details on setting part properties). Ordinary behavior is restored by typing “0”, or by restarting Pick Face.



Distance Measurement

Use the Distance Measurement feature to find the distance between two points:

- Start or restart measure mode by pressing **[m]**.
- Click the starting point.
- Move the mouse pointer and click again to establish the second point along a “ruler”.
- TenGun “snaps” the points you selected with the mouse to the nearest points in the actual points in the data and displays the distance.

Keys: **[m]**. Press **[Esc]** to turn off measurement.

Mouse: Drag the handles of the ruler to move the two endpoints.

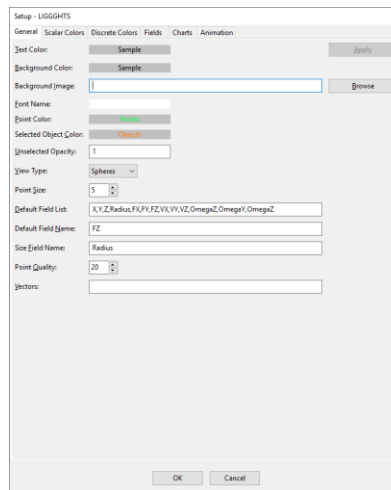
Profiles

A "Profile" contains information describing Field properties such as the data type (e.g., integer, float), labels for scalar bars, and coloring information.

Menu: Edit → Settings.

Toolbar: .

General Settings



Settings are:

- Text Color. The color for text messages. Double-click the color to display a Color Picker Dialog.
- Background Color. The background color for text messages. As above, double-click the color to change it.
- Background Image. The name of an image file that can optionally be added to Time Series displays. Whether or not this is actually used is determined by an option in the File Processing Options dialog.
- Font Name. The name of the font to use. Most TrueType™, OpenType, and FreeType fonts can be used. (Some of these do not work with the graphics library, so you may have to experiment a bit.) Double-click the font name to change it.
- Point Color. This is the color to use for points when a color table is not assigned.
- Selected Object Color. This is the color to use for selected vertices, edges, faces, and objects.
- Unselected Opacity. When an object is selected, the opacity of the other objects can be changed. This allows you to see the selected object through any containing objects. The value range is 0.0 for fully transparent to 1.0 for fully opaque.
- View Type. Sets the default point view type such as Points or Spheres.
- Point Size. Sets the default point size from 1 to 99, with 1 through 10 being common.

- Default Field List. This list defines the names of fields for simple data files that do not contain any field definitions. Field names are separated by commas.
- Default Field Name. If this item is set, TenGun will automatically select this field whenever you open a file using this profile.
- Size Field Name. TenGun supports variable point size display which is useful for displaying Discrete Element Method (“DEM”) and Molecular Dynamics results. The Size Field Name is the name of a field in the output data containing the particle size, for example “Radius”.

To display such data, TenGun converts individual points into spheres of the associated radius. Additionally, TenGun clones properties from each particle such that if you do Point Picking anywhere in the sphere, it will report the information for the original field data. The “Point Number” displayed will no longer match because there are now more points. However, if the input data has an “ID” field, the original ID will be correct no matter where you click on the sphere.

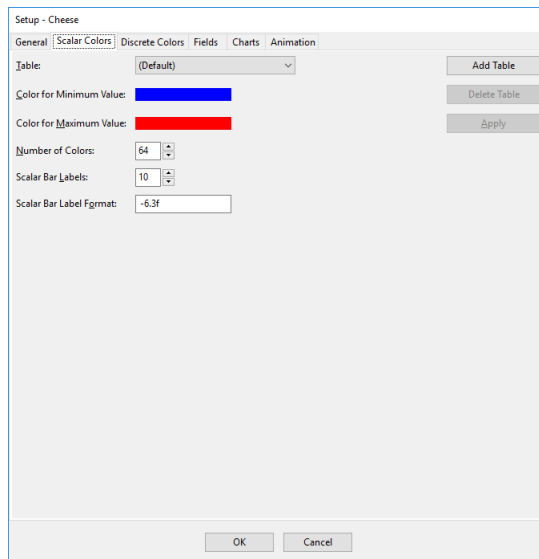
This feature can generate a significant amount of additional data, so it is not intended to be used with huge point clouds.

- Point Quality. The value sets the number of subdivisions used to create the graphic representation of each sphere. A setting of about 5 ~ 10 is not beautiful, but reasonable when you are just verifying your analysis run. Then, before for making presentation material you can raise this to a higher value such as 20. Note, however, that this setting only takes effect for newly loaded data, so you cannot change it for a scene that is already being displayed. If the Point Quality is set to 0, TenGun uses the normal Point Size setting to display a scene, and ignores the “Size Field” setting. Finally, for higher quality output, you can display the scene using a Solid View instead of the usual Sphere or Point View used for particle data.
- Vectors. Vector definitions for Vector View mode. Refer to [Vector View](#) for details.

Continuous Scalar Colors Settings

Scalar Color settings are used when displaying the Scalar Bar for continuous variables such as Density or Pressure.

Note: The Scalar Bar Labels and Scalar Bar Label Format on *this* screen apply to the *Default* Scalar Color Table. The equivalent settings for the non-default case can be made on a *field-by-field* basis in the Fields editing screen.



Settings are:

- **Table.** Used to select a specific Scalar Color Table. Use the Add Table Button to add a new table, and use the Delete Table Button to delete the selected table.
- **Color for Minimum Value.** Scalar colors are based on a *color wheel* concept. Red, Green, and Blue are at 0, 120, and 240 degrees respectively, wrapping back to Red at 360 (0) degrees. The "low" and "high" colors therefore select the starting and ending points on the wheel. Double-click the color to display the Color Picker Dialog.
- **Color for Maximum Value.** The "high" color as explained above. Double-click the color to change it.
- **Number of Colors.** Sets the total number of colors. A color gradient with this many entries (e.g., 64 colors) will be created for values between the minimum and the maximum.
- **Scalar Bar Labels.** Sets the number of labels to display on the Scalar Bar when using the *Default* Scalar Color Table.
- **Scalar Bar Label Format.** This specifies how floating-point values are to be displayed. See [Scalar Bar Label Format](#) below.

Scalar Bar Label Format

A format specifies how floating-point values are to be displayed, and must be of the form:

[flags][width][.precision]specifier

Users familiar with "C" will notice that this is a "printf" format string *without* the % sign.

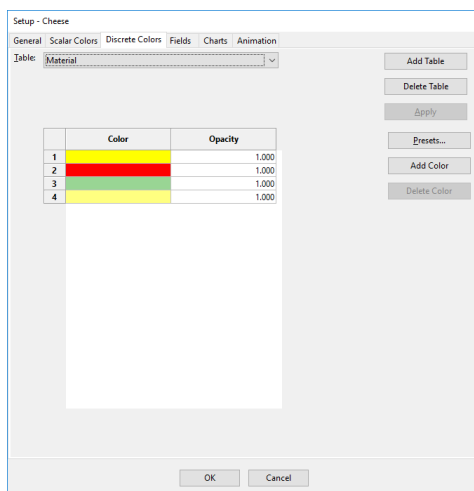
where:

- "flags" are optional modifiers. "+" forces the display of a plus sign for positive numbers, and "-" causes the output to be left-justified.
- "width" specifies the *minimum* display width. If the formatted string is shorter than this it will be padded with spaces.
- "precision" specifies the number of digits to display after the decimal point.
- "specifier" specifies a style. Styles can be: e (scientific notation), f (decimal floating-point), or g (best fit). "Best fit" means use floating-point format if there would be many trailing zeros, otherwise scientific notation.

The default format is "-6.5g".

Discrete Colors Settings

Discrete Colors Tables are used when there are a limited number of possible values for a field. For example, an analysis model typically has only a few different materials. In this case you can use a Discrete Colors Table. This maps, one-to-one, each material to a different color. Materials are then easy to distinguish during visualization.



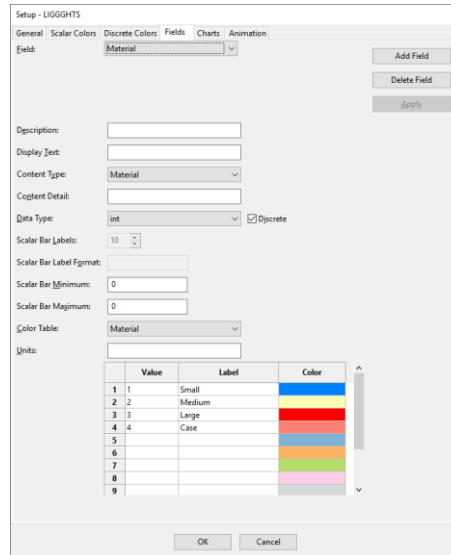
Settings are:

- Table. Used to select a specific Discrete Colors Table. Use the Add Table Button to add a new table, and use the Delete Table Button to delete the selected table.
- To add a predefined color set, click the Presets Button and select one of the many predefined color tables.
- To add colors individually, click the Add Color Button.
- To delete a color (1) select it by clicking the color number in the left column, and (2) click the Delete Color Button.
- To change a color, double-click on it. The Color Picker Dialog will be displayed.
- Opacity. Values can range from 0.0 (totally transparent) to 1.0 (totally opaque). You may be able to obtain some interesting display effects for your scene by changing this value. For example, if you have a scene with water and gas bubbles, you can lower the opacity for the water so that the gas bubbles are more visible. Double-click a value to change it. Then press **Enter** to complete the edit, or **Esc** to cancel.

In this sample screen we have already added a Field named “Material” on the Fields tab, and here in the Discrete Colors tab we define the color scheme for material 1 through 4.

Field Settings

This screen is used to define the details of data fields in your input files.



Settings are:

- **Field.** This dropdown list contains the names of available fields. The sample profiles contain a number of common fields such as AX, AY, and AZ for acceleration, VX, VY, and VZ for velocity, and so on. Use the Add Field Button to add a field, and use the Delete Field Button to delete a field.
- **Description.** An optional description of the field.
- **Display Text.** If this is set, this text will be displayed in various places instead of the field name. For example, if you have a field named "P", you might want to actually display "Pressure".
- **Content Type.** The selected value indicates how this item is related to the model, for example, the data is related to "Acceleration", "Velocity", "Stress", etc. This is currently unused, but may be used in a future version to provide additional analysis capabilities.
- **Content Detail.** This specifies the English name of the content. It is currently used in handling variable particle sizes so that we can determine that the field named "R" is actually a radius, or the field named "Phi" is a diameter. Allowed values are "Radius" and "Diameter".
- **Data Type.** Specifies how the data should be stored internally. Possible settings are:
 - **byte.** 8-bit unsigned integer. Range: -128 to +127.
 - **double.** 64-bit floating-point. Range: 2.225 E^{-308} to 1.797 E^{308} with approximately 15 digits of precision.
 - **float.** 32-bit floating-point. Range: 1.175 E^{-38} to $+3.402 \text{ E}^{38}$ with approximately 7 digits of precision.

- int. 32-bit signed integer. Range: -2,147,483,648 to +2,147,483,647.
- long. 64-bit signed integer (in "C" this is really called a "long long"). Range: -9,223,372,036,854,775,808 to +9,223,372,036,854,775,808
- short. 16-bit signed integer. Range: -32768 to +32767.
- string. A character string in UTF-8 format. Currently of limited use in TenGun.
- uint. 32-bit unsigned integer. Range: 0 to 4,294,967,295.
- ulong. 64-bit unsigned integer. Range: 0 to 18,446,744,073,709,551,615.

To conserve memory you should try to use Data Types that are as small as possible but still contain the range of your values.

- Discrete. This option can be used for fields that contain a limited number of different values. (Up to about 10 is reasonable). This option instructs TenGun whether to use Scalar or Discrete Colors Table.
- Scalar Bar Labels. Number of labels to display on a Scalar Bar for this field. Please refer to [previous discussion](#).
- Scalar Bar Label Format. The display format for this field. Please refer to [previous discussion](#).
- Scalar Bar Minimum. TenGun normally uses the actual minimum and maximum field values to set the Scalar Bar range. You may set fixed limits by setting the minimum and maximum (see below). To reset the normal behavior, set both values to 0.
- Scalar Bar Maximum. Maximum value for the scalar bar range as explained above.
- Color Table. Use this to select a particular Color Table to apply to this field.
- Units. You can set the units to display on the Scalar Bar for this variable such as "m/s" or "MPa".

When displaying a Time Series, you may temporarily change the minimum and maximum values by pressing **S** **E**. These settings are not saved in the profile.

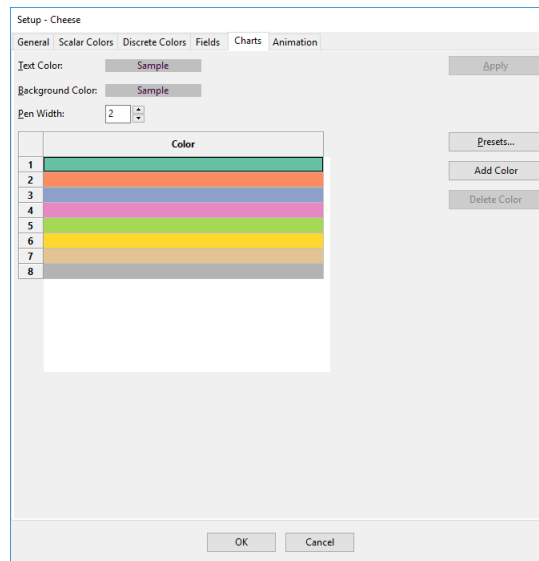
In this screen we have defined a *discrete integer* field named "Material", and have added labels for each of four possible values.

For discrete variables you can specify value/label pairs. The value depends on your particular data, and will be a field value such as a Material number. The corresponding label is displayed on the Scalar Bar. Double-click a label or value to change it. Press **Enter** to complete the edit, or **Esc** to cancel.

When a discrete variable is a floating-point type, the meaning of the "value" changes. Rather than being a fixed value, it becomes an upper limit. For example, for a stress field, you might set values of 0.0 (lower limit), 1000.0,

2000.0, and 3000.0 (upper limit) to define four ranges that are used to select one of four colors from a Discrete Color Table. Where this becomes interesting is if you adjust the opacity in the color table. For example, to make high-stress areas easy to see, set the opacity to lower values (e.g., 0.1) for stress levels that you are not interested in, and set the opacity to a high value (up to 1.0).

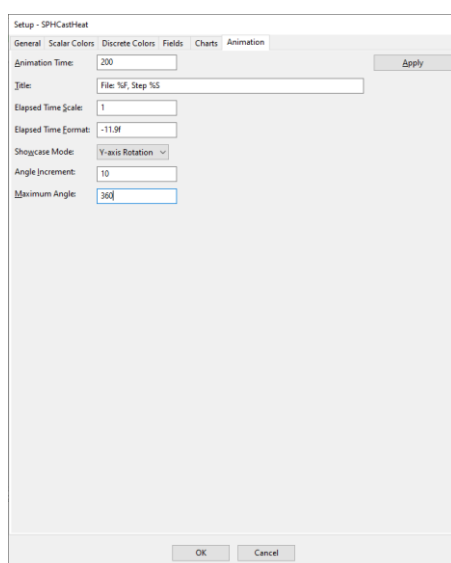
Chart Settings



Settings are:

- Text Color. Color for labels in the chart.
- Background Color. Chart background color.
- Pen Width. Width of lines in a chart.
- Color Table. One color for each possible variable. If there are more variables than colors, colors will be reused starting from the beginning of the table.

Animation Settings



Settings are:

- Animation Time. The number of milliseconds between animation frames.
- Title. Text to display in the animation screen. The text may include the following macro values:
 - %D Current date.
 - %E Elapsed time.
 - %F File name.
 - %P Number of points (particles).
 - %S Animation step number.
 - %T Current time.
- Elapsed Time Scale. TenGun multiplies the time step number by this scale value to arrive at the elapsed time. In many simulations, one output file represents the results after a number of simulation steps. For example, the time step may be 1 microsecond, but the data is written only every 1000 steps. In this case the time scale value would be set to 0.001 (0.000001 seconds x 1000 steps).
- Elapsed Time Format. This is a format string that you will want to adjust depending on the Elapsed Time Scale. For example, for the above Elapsed Time Scale example, the smallest time will be 1 millisecond. In that case you can set the format to “-4.3f” which will display 3 digits after the decimal point. Refer to [Scalar Bar Label Format](#) for format string details.
- Showcase Mode. After displaying a time series you can rotate the camera around to showcase the final product. The mode can be either “None”, “X-axis Rotation”, or “Y-axis rotation”.

- **Angle Increment.** Showcase Mode increment in degrees for the rotation. A positive value moves the camera counterclockwise, and a negative value moves clockwise.
- **Maximum Angle.** The maximum angle for Showcase Mode. This will normally be 360°, but can be less for a partial rotation, or more if you want to rotate several times.

Help

The following information is available from the **Help** menu item:

- Documentation. Displays this file.
- Quick Reference. Displays keyboard and mouse usage information.
- Release Notes. This contains program update history and known limitations.
- About This Scene. Displays metadata for the current scene.
- About. Displays TenGun program and license information.

Additional Documentation

The following documentation is also available in installation folder under the Docs subfolder:

- EULA-eng.rtf. The End User License Agreement.
- ThirdPartyLicenses.pdf. This file contains license information about Open Source libraries used by TenGun.

Assembly Mode (Pre-Processing)

TenGun can create input data for various analysis applications. This typically involves converting multiple CAD files to particle format, and then adding application-specific parameters such as timing control, material properties, boundary conditions, and so on. TenGun encapsulates these pre-processing activities with a few simple operations.

Pre-processing features include:

- Read a number of CAD format files including IGES, OBJ, PLY, STEP (ISO 130303-203 and 130303-214), and STL.
- Convert CAD models to analysis-application input format.
- Convert a CAD mesh to a point cloud.
- Edit CAD object geometry.

Assembly Mode usage is explained in the remainder of this chapter.

NOTE WELL: Assembly Mode requires a “Professional” license. If you do not have such a license you will still be able to explore features related to Assembly Mode. However, you will not be able to save any related files.

Generating Data

the software TenGun can create some basic object shapes that can be used to create an analysis model. To do so:

- Open an Assembly view by creating and empty view with **File**→**New Assembly** or by opening a CAD file in Assembly Mode.
- Insert one or more objects from the Object menu. Available object types include:
 - Box (Cube). The box may have either one or two open faces.
 - Cone/Frustum. Creates a cone or frustum.
 - Extrusion. Create a 3D object by extruding a 2D surface. See [Revolved and Extruded Surfaces](#).
 - Pipe/Circle. An open-ended cylinder. You may optionally add caps to either end. Setting the pipe length to 0 will create a 2D circle.
 - Revolution. Create a 3D body of revolution from a 2D surface. See [Revolved and Extruded Surfaces](#).
 - Solid Plate. You will be prompted for the dimensions. One of the dimensions can be 0 to create a 2D plate.
 - Solid Disk. You will be prompted for the dimensions. Setting the depth to 0 will create a 2D disk.
 - Sphere. A spherical object (hemisphere by default).
 - Torus. You can create a full torus by specifying an Angle of Revolution of 0° or 360°, or a partial torus by specifying some angle in-between.
 - Wedge. Creates a solid wedge or pyramid-shaped object. The X Min and X Max values specify the top left and top right vertices of the face in the X plane. Similarly, the Z Min and Z Max specify the top front and top rear vertices of the face in the Z plane. If all of the X, Y, and Z-related values are the same the result is a cube. Changing these values therefore allows you to generate a number of different shapes. For example, you can create a pyramid by setting X Min and X Max to half the width, and Z Min and Z Max to half the depth.
- Duplicate. Creates a duplicate of each selected object. Duplicated objects are inserted to the right of the original; from there you can move them as required.
- In addition to inserting objects as described above, you can open a file and append it to an existing view.

Keys:

n: Next object.
N: Previous object.
s: Toggle selection of all objects.
Del: Delete object.
Ctrl+N: New Assembly.
Ctrl+G: Edit Geometry.

Menu:

File → New Assembly.
Object → Insert Object.
Object → Duplicate.
Object → Delete Object.
Tools → Difference.
Tools → Intersect.
Tools → Union.
Tools → Edit Geometry.

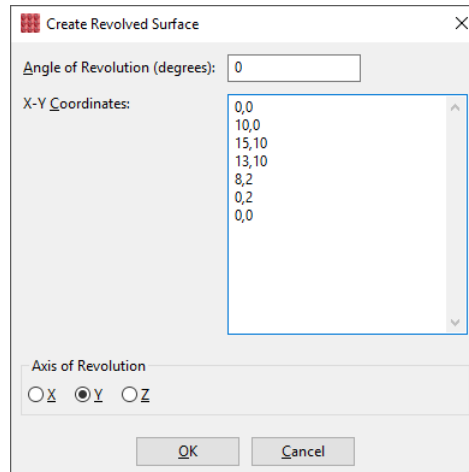
- The default size of each object is such that the length, width, and/or height are 100mm. This is a practical size for a part that you might analyze with millimeter-sized particles.
- For many objects you can set a *wall thickness* that by default will be a few particles thick. For round objects such as a Pipe, the wall thickness is implied by the inner and outer radius. The wall thickness concept is used to control the strength of a fixed boundary versus free-flowing particles in an analysis.
- To create objects that are completely full of particles and do not have empty space inside:
 - Box or Cone: Set the wall thickness to 0.
 - Pipe, Sphere, or Torus: Set the inner radius to 0.

With these basic shapes you can create a number of objects that are handy in particle analysis such as:

- A box or pipe with openings can be used as a particle container, and a plate can be used to compress them.
- A solid box might be used to represent ice in a liquid.
- Torus segments can be used as a rounded pipe segments.
- A frustum with open ends can represent a funnel. The funnel can contain material that is injected into some other part of the system. This case can be used, for example, to represent a 3D printer head, a medical device, or a portion of a die casting system.
- Basic shapes can be combined with [Boolean Operations](#) to create more complex objects.

Revolved and Extruded Surfaces

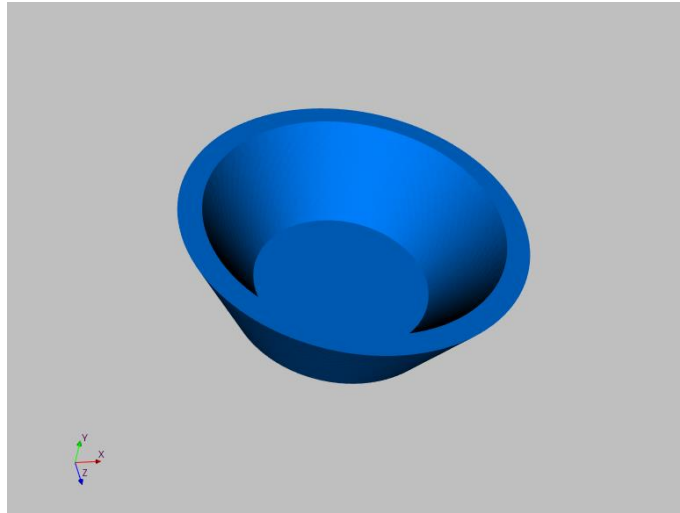
To create a body of revolution from a 2D surface, select **Object** → **Insert Object** → **Revolution**. The following dialog will be shown:



Fill in the values:

- **Angle of Revolution.** The number of degrees to rotate the surface. An angle of 0° is treated as 360° .
- **X-Y Coordinates.** Input the X-Y coordinate pairs for the surface, one per line, with values separated by a space or comma. You need at least three pairs to create a surface. An unclosed surface will result in a shell being created.
- **Axis of Revolution.** Select the axis around which to rotate the surface.

The above input will create a bowl:




Creating an extruded object is nearly the same. Select **Object**→**Insert Object**→**Extrusion**. A dialog similar to the Create Revolved Surface dialog will be shown (omitted here to save space). The main difference is that you must input the extrusion *length* rather than an angle.

Part Colors


To change the coloring of assembly parts, create a [Discrete Colors](#) table named “Assembly”. The first part inserted uses the first color, the second part the second color, and so on. If there are more parts than colors, TenGun cycles through the table from the beginning.

The dialog for extrusion is quite similar and not shown here, except for extrusion you input the extrusion length instead of an angle.

Exporting Data

 This section describes the two ways that TenGun can export data.

Export Model as Point Cloud

 This feature can convert solid models in the current scene to a point cloud. Any existing point cloud data in the scene is ignored during the conversion.

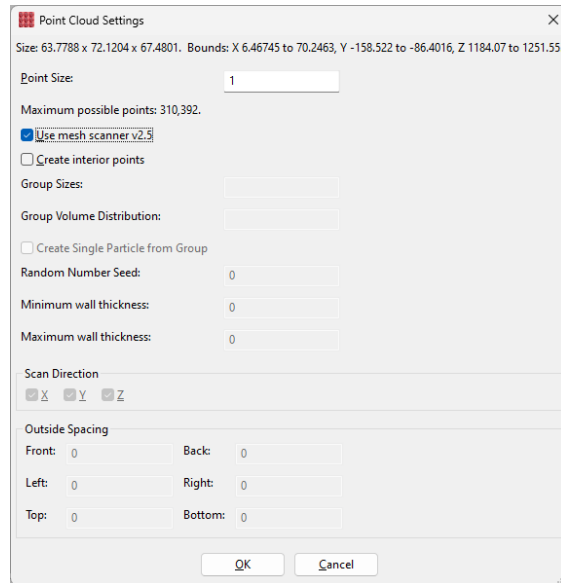
Menu: File → Export → Model as Point Cloud.

Experimental Feature

To create a point cloud of the model itself you only need to set the Point Size. A point cloud is created where points fill all of the model's surfaces.

The “Use mesh scanner v2.5” option attempts to cover all surfaces with particles. The legacy algorithm is similar, but it tries to identify solid walls, ignoring the empty space between those walls.

The newer algorithm works better for objects that do not have explicit solid walls such as OBJ and STL meshes.



The following settings allow you to create particle *groups* that can represent clumps of materials of different sizes:

- **Group Sizes.** This is a list of one or more cluster sizes separated by spaces or commas. When set, the Point Size value becomes the minimum grid size, and each of the values in this list is the diameter of the cluster. For example, if the Point Size is set to 100 (perhaps in microns), the Group Sizes might be 100, 120, and 140. Values must be \geq the Point Size.
- **Group Volume Distribution.** If Group Sizes is set, this field contains the desired volume distribution for each of the groups (e.g., 25, 50, 25) expressed as a percent of the total volume. Values are comma or space separated.
- **Create Single Particle from Group.** When unchecked, a “group” will consist of one or more particles of size “Point Size” organized as a sphere. This organization is suitable for SPH analysis. When checked, the group is output as a single sphere, which is more suitable for DEM analysis.

- Random Number Seed. Particle groups are placed randomly. If you do not specify a Random Number Seed, the same input will generate the same output. If you actually want to generate a different distribution, set this to a different seed. Remember to make a note of your seed values so you can reproduce the same output if necessary.

When generating particle groups, TenGun creates the Point Cloud file for the entire model, and one additional Point Cloud file for each Group Size. The individual files can be imported back into an assembly, and then exported once again when creating Analysis Data.

For simple objects, the default wall thickness settings are usually appropriate. Complex objects, however, may require some experimentation with these settings:

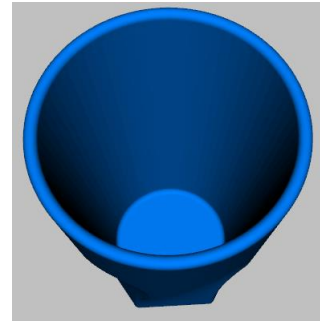
- Minimum wall thickness. When this value is non-zero, the algorithm will ignore short solid segments so that the segment will not be treated as the start of a wall. While scanning a cylindrical object, for example, most of the time the scan detects going inside the cylinder, some empty space, and then the other side of the cylinder. While scanning the extremes of the cylinder, however, the scan never actually goes *inside*. Note that the units are *points*, so you may need to convert the wall thickness in your model (e.g., millimeters) to points to find a good value.
- Maximum wall thickness. This is similar to the above, but instructs the scanner to discard false walls. A false wall can occur if there are overhanging portions of the model such as flanges.
- Scan Direction. The scanning needs to find wall entrance and exit points in order to decide if there is empty interior space. By default, the scanner does this in three dimensions. In some cases, you can scan in only one direction to speed up the point cloud creation. For complex objects, you can sometimes improve the output by scanning in only one or two directions. (See examples below.)

Output quality is also affected by the quality of the input.

Finally, for interior point clouds you can set the Outside Spacing to shave off the specified number of particles in the specified direction(s). Use this option to prevent filling the object entirely. For example, if you have a container of liquid, you may not want to fill it exactly to the top, so you will set the “Top” value to several particles. This option is also useful in particle analysis to prevent particles from initially being too close to walls.

Point Cloud Interior – Simple Example

This cup example works well with interior point creation. The cup is actually upright, but has been rotated so that you can see the inside.

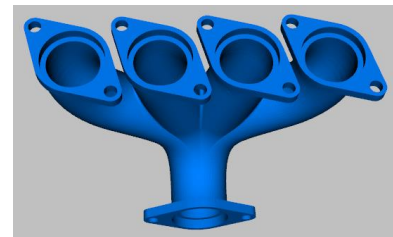


Scanning in the all three directions works with no problem. Note, though, that the Y-direction scan does not detect anything because it only finds one wall. Therefore, the scan could actually be done only in the X or Z direction.

Also, this is a perfect example of where “Outside Spacing” can be used. Setting the “Top” value prevents the cup entirely to the brim.

Point Cloud Interior – Complex Example

This exhaust manifold is a nightmare:




- We don’t want to include the holes in the flanges as interior space, but scanning horizontally this case looks like wall-empty-wall.
- Scanning vertically, the flanges look like the start of a wall, but we don’t want to treat them as walls.
- We don’t want to treat the empty horizontal space between the upper pipes as interior space.
- You can’t see it here, but the upper pipes are tapered. They are thicker near the flanges than near the middle.

The following steps allow you to create a reasonable (although not perfect) approximation of the interior:

- Cut off the flanges with the 3D Cut function. The flanges are probably irrelevant to the analysis anyway.
- Scan only in the Y and Z directions. This avoids a false wall problem between the pipes in the X direction.

The result leaves some slight artifacts caused by the tapering of the pipes—a challenge left for a future version.

Export Analysis Data

 This feature combines all of the objects on the current panel to create input data for a third-party analysis program. When opening the files, pay careful attention to the "Load as separate parts" Open option as this will affect the number of generated parts.

Operation is as then follows:

- Apply part properties to each part in the scene:
 - Select an object. If no part has been selected, the first part will be selected automatically.
 - Invoke the Edit Part Properties function. The first you will be prompted for an "Export Template".

Keys:   ("Edit Part" properties).

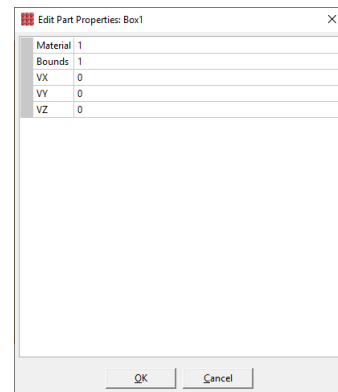
Menu:

Object → Edit Part Properties.

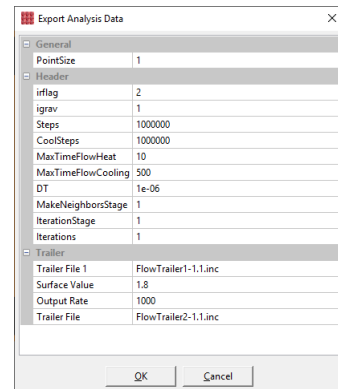
File → Export Analysis Data.

The template is specific to the analysis program you wish to use. The standard installation includes only a sample template "SampleApp.export" that you can use to learn how to use this feature. Please contact us for information about templates that we have developed or about how to create your own template.

- Set the appropriate property values. Again, this is specific to the target application, but typically includes settings for the material type, boundary condition, velocity, static load, and so on.
- [Experimental Feature](#). You can use Edit Part Properties on faces and edges. Use Pick Faces (**[p]** **[f]**), Pick Edges (**[p]** **[e]**), or Pick Vertices (**[p]** **[v]**) to select a sub-object. Then use Edit Part Properties to set the values. This feature is useful, for example, to set a static load in a particular area of a part.



- Select **File** → **Export** → **Analysis Data**. In the dialog that is displayed, fill in the settings shown for the specific template:
 - General. This includes general settings for the target application, and is where you set the desired particle size.
 - Header. Contains values appearing at the beginning of the generated data file.
 - Trailer. Contains values appearing at the end of the generated data.



Part-specific data is written between the Header and Trailer sections. This data typically includes the coordinates of each particle, velocities, material type, boundary conditions, and other application-specific values.

- Sometimes you have tables that are fairly constant such a table of Densities and Young's Modulus values. In this case you can use the template processor's "file" data type to insert these tables into your output.
- When you select **[OK]**, TenGun merges a point cloud for each input object (part) with your export template and part property settings into a single output file.
- Optionally perform manual editing on the output file. This may be required if the template editor cannot automatically generate 100% of the output.
- Run the analysis application.

Hints

- Your CAD model and particle size must use the same world coordinate dimensions. For example, if you have a 10cm pipe and will have millimeter

particles, the pipe length should be 0.010 (meters), not 10 (centimeters). If necessary you can rescale CAD objects using the [Geometry Editor](#).

- The most recent General, Header, and Trailer settings are saved in a file named “*export_template_name*.export.config”, located in the “C:*user_name*\AppData\Roaming\TenGun” folder.

Assembly Example – Part 1: Assemble the Parts

This section is a step-by-step example of assembling various CAD parts, creating particle data to fill the inside, use of the [Geometry Editor](#), and exporting the data for use in an analysis application.

For this example, we will make a contraption for the person who has everything: a “Swiss Cheese Hole Filler”. This is a totally useless device, but it is fun and easy to implement.

Here is our strategy:

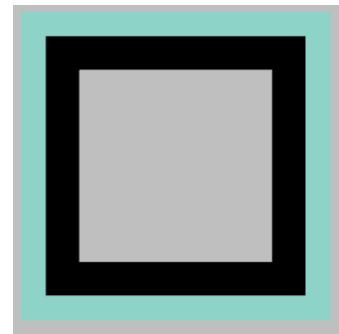
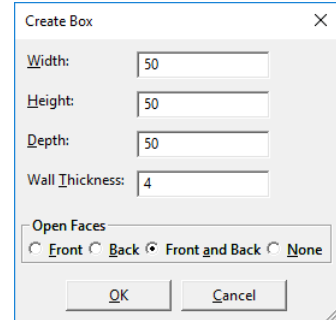
- We want to make a device to push particles into this block of Swiss cheese. The cheese data was a sample STEP file found on the Internet. We know from inspecting the cheese in the [Geometry Editor](#) that the size is 50.8 x 50.8 x 50.8mm.
- We need a container to hold the particles, so we'll use TenGun's box object. We also need a square plate that fits in the box. The plate will be used to push the particles.
- We need some particles to fill the box.
- We'll create the objects in this order: Box, Plate, Particles, and finally cheese. The reason for this sequence is that when we create the interior particles for the box, we don't want TenGun to pre-fill the cheese, too.



Create the Box

Here we will use TenGun's built-in box object as a particle container. As an alternative in a real world example, you can import your own CAD model as a substitute container.

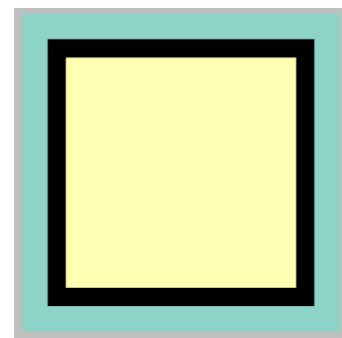
- Create a New Assembly scene (File → New Assembly). For this example you can use any Display Profile. This selection only affects the colors that are used.
- Create a Box (Object → Insert Object → Box). Notice that the box size is set to the same size as the block of cheese. (We want at least one dimension to be the same.) Also notice that the "Front and Back" will be open. One of the open sides will be connected to the cheese, and we will insert a plate in the other side.
- The Scene Panel now shows the box like this.



Create the Plate

Create the plate with:

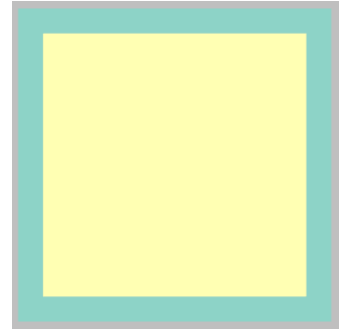
- Object → Insert Object → Solid Plate
- Set the dimensions of the plate to 42 x 42 x 4. 42 is 50 – the thickness of two walls.
- Select **OK**, and the Scene Panel will look like this. (It is difficult to see here in 2D, but the black portion is a shadow, and the plate is inside the box.)



Align the Box and Plate

We now have a box and a plate, but the plate is located at the center of the box. To align the plate to the outside of the box:

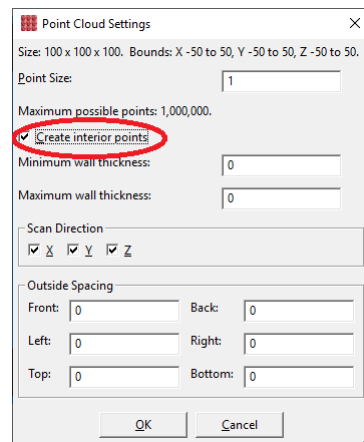
- Type **p** **f** (Pick Faces).
- Click the plate. It will be highlighted.
- Click an outer *edge* of the box. The front outer wall of the box will be highlighted.
- Type **j** to join the objects. The front face of the plate will be aligned with the front of the box.
- The Scene Panel will now appear like this. If you rotate the scene to look from the back, though, you can see that the plate is indeed aligned with the front of the box.



Create Cheese Particles

To create particles to fill the box:

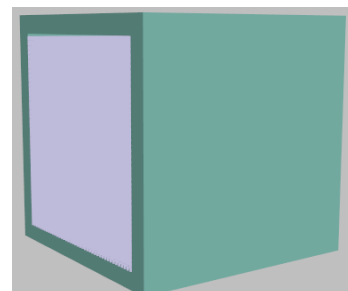
- **File** → **Export Point Cloud**.
- Set the Point Size. Remember this value because you will need it again later.
- At this stage don't want to convert the box and the plate into particles. Rather, we want to create particles to fill the inside of the box. Therefore, check "Create interior points".
- Select **OK**.
- Enter a file name to write the particles to. We've used "CheeseParticles.csv" here.
- TenGun will create the particle data.



Insert Cheese Particles

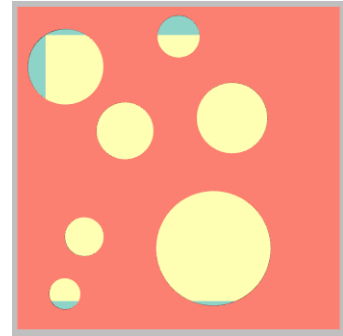
Insert the particles into the box/plate assembly:

- **File** → **Open**.
- Choose "CheeseParticles.csv". Be sure to check the "Append to current scene" option.
- The scene, if rotated so you can see the back, will look like this.



Add the Cheese

- File → Open.
- Choose “Cheese.step”. Be sure to check the “Append to current scene” option.
- The Screen Panel will appear like this. As you can see, the Cheese object is on top of the box, which is not what we want. We’ll fix this in the next step.

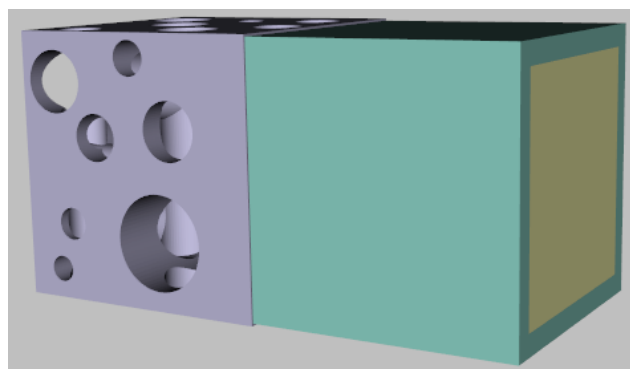


Move the Cheese

Use the [Geometry Editor](#) to move the cheese off of the box:

- Make the Cheese object the active object. Since we have four objects in the scene, you can press “n” to go forward three objects (skip the box, plate, and particles) to the cheese, or you can press “N” to go backwards from the box to the cheese. Going either way, you should end up with the cheese being highlighted.
- Press **Ctrl+G** or use **Object → Edit Geometry** to open the Geometry Editor.
- Recall that the box size is 50x50x50. Move the Cheese object 50 units in the -Z direction using the Translate function so it is just outside of the box. In other scenarios you may need to rotate something, or you may want to use [Object Alignment](#) to join faces.

At this point the scene, rotated, should look like this:



Set the properties of each part as required by the analysis application. One by one:

- Select a part (right-click).
- Invoke the Part Properties edit screen (type **e** **p** or use the **Object** → **Edit Part Properties** menu item). We are not really going to run analysis in this tutorial, so you can pick any available Export Template.
- Fill in the appropriate parameters. In this example, each part must be given a unique value for the Material type, Bounds (boundary condition). Additionally, the VZ (Velocity-Z) value must be set for the moving plate.
- Select **OK**, and repeat for the other parts.

In this example, the full assembly contains particles that are hidden inside the box, so you cannot right click them to select them. In this situation you can select the part in the Tree View. You can also type **n** or **N** to cycle through the parts until the selected object is not visible (because it is inside the box), and *then* invoke Edit Part Properties.

Hints

- It usually takes a few tries until the data is perfect. As you add objects to the scene and move them around, use **File** → **Save Assembly** to save your work. Then if you need to re-do something you can reload your previous data and modify it.
- Notice that I have mentioned the *rotated scene* several times in this example. An alternative and perhaps better way of assembling the parts would be to attach the box to the *right-hand* side of the cheese instead of the front. The box must then be initially rotated so the open front and back are oriented correctly, but that is perhaps easier.

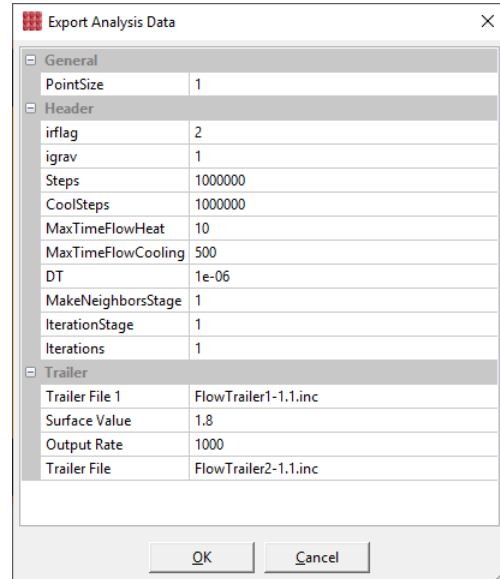
Menu: **File** → **Save Assembly**.

This saves CAD objects only. It does not save particle data.



Assembly Example – Part 2: Create Analysis Data

Next, we would like to run our cheese press contraction through an analysis program. To do so:

- Select **File** → **Export Analysis Data**.
- Fill in the parameters required by that application.
- Select **OK**.
- TenGun converts the Box, Plate, and Cheese Block to particle form, and also include the “cheese particles” that we previously created.
- Enter an output file name when prompted. TenGun will then write all of the particle data and analysis application parameters to that file.
- In some cases you may want to perform some manual editing or fine-tuning on the output.
- Run the analysis.



Object Editing Tools

Operations shown with the Time Series Mode  icon operate only on data that originated as a Point Cloud. Operations shown with the Assembly Mode  icon can be used with STEP and IGES data, and with STL data that was read with the extended reader.

Analyze Mesh

Display information about the various polygons types in a mesh. This provides useful information that can be used with mesh subdivision features. Output is shown in the Message Window.


Analyze Point Density

Calculate the point density for the specified search radius. TenGun then finds the number of neighbors for each point within that radius, storing the values in a field named “Neighbors”.

Be careful to set the radius to no more than the particle size x 2 or 3. Otherwise the search will include too many points and will take a long time.

This feature is useful for spotting flaws in a model such as pores and cracks which will have low-density values.

Background Mesh

 Some time series data (e.g., from LIGGGHTS) consists only of particles without a container. You can enhance to display by adding background meshes that represent the container and moving plates. Property values of the mesh files are ignored.

There are three options on the **Background Mesh** submenu:

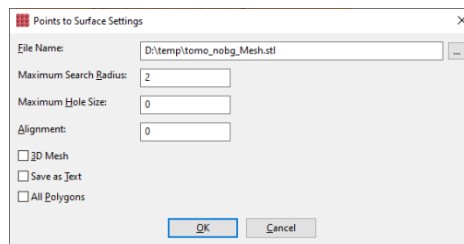
- **Add**. If a single file is selected it will be appended to any existing meshes in all scenes. If multiple files are selected, each file’s mesh will be appended to the existing mesh for that scene.
- **Replace**. Same as above, but previously loaded meshes are replaced.
- **Remove**. Removes background meshes.

Points to Surface

🕒 This feature applies a Delaunay 2D or 3D to a point cloud, allowing you to create a mesh from a set of unstructured points.

Usage is as follows:

- Select the **Tools** → **Points to Surface** menu item. The following dialog will be displayed:




- Fill in the parameters:
 - File name. Enter the file name here, or browse by selecting the button. Supported output formats are OBJ, PLY, STL (default), VTK, and VTP.
 - Maximum Search Radius. If set to zero, the filters output all triangles (2D case) or tetrahedra (3D case). If non-zero, output is limited to objects that fall within a bounding sphere defined by this radius. In the latter case, vertices, lines, and triangles can be generated. However, TenGun suppresses the output of vertices and lines.
 - Maximum Hole Size. Holes in the mesh up to this radius are removed by a hole-removal filter.
 - Alignment. If non-zero, particles are aligned so that the coordinates are all multiples of this value, and redundant points are removed.
 - 3D. When checked apply a Delaunay 3D filter, otherwise a Delaunay 2D filter. When you apply a 2D filter to 3D data, the filter ignores the Z-coordinate and creates a mesh of the surface. This is useful, for example, to create a lithophane.
 - Save as Text. By default, output files are written in binary format. Check this option to create ASCII text files instead.
 - All Polygons. By default, the generated mesh will consist of triangles. If this option is checked and the output file type is either VTK or VTP, higher-order polygons such as tetrahedral will be written. TenGun currently does not support displaying the resulting “unstructured grid” files. However, you may use the output with other tools to extract the polygons.

- Select **OK** to start conversion.
- The converted mesh may be saved in another format such as a STL, VTK, or VTP using the **File → Save Point Data** menu item. This data can then be used as input to another process, and even used for 3D Printing.

Notes

- For 3D, the tool is currently limited to moderately-sized input data up to a few hundred thousand points because the conversion process can become slow and use a lot of memory.
- For 3D, you will generally get good results if the Maximum Search Radius setting is approximately 2 to 3 times the point spacing. If you use a larger value, distant corners of the object can be connected, and you can end up with a useless giant blob.
- The Delaunay algorithm is likely to fill in small holes. To preserve the holes you must provide particle input data with finer spacing.
- The Delaunay algorithms work best with irregularly shaped objects. For example, conversion of simple plates or cubes is very slow.
- Very thin areas do not convert well with Delaunay 3D. Thin areas need to be a few particles thick. Otherwise these areas are likely to become holes. A trick that works in some cases is to rotate the object slightly so that the flat portions are perfectly parallel to a major plane.
- During conversion from a point cloud to a mesh, scalar values such as temperature and pressure are lost.

Remove Outliers

 This feature removes “rogue particles”, that is particles that have leaked from the simulation container. Such particles can interfere with the display because particles at extreme positions require that the camera be very distant in order to view the entire model.

When this feature is selected you will be prompted for the Minimum Neighbors and Search Radius (a multiple of the particle size). Any particle that does not have at least “Minimum Neighbors” particles within this radius are removed.

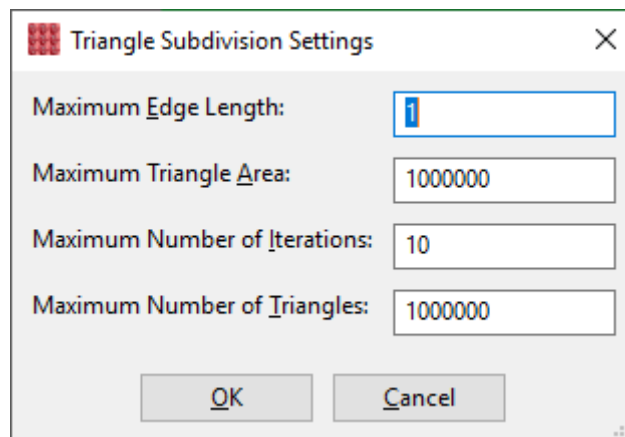
The Remove Outliers function cannot be undo (you need to reload the data to get back the original).

Subdivide Triangles

🕒 This feature first converts non-triangles to triangles. Then, all triangles are subdivided according to the specified criteria, resulting in a triangular mesh with more detail than the original.

Usage is as follows:

- Prepare a mesh. You can read this from a CAD file, or create a mesh via the **Tools** → **Subdivide Triangles** menu item. The following dialog is displayed:



- Fill in the parameters:
 - Maximum Edge Length. Edges that are longer than this value will be divided in half, and associated triangles regenerated.
 - Maximum Triangle Area. Triangles with an area larger than this value will be subdivided. This option may sometimes introduce holes into the mesh.
 - Maximum Number of Iterations. The subdivision process is iterative. You may limit the number of passes by setting this value.
 - Maximum Number of Triangles. You may limit the number of triangles generated. However, if the limit is reached, the result mesh may contain holes.
 - Select **OK** to start conversion.

Object Alignment

⚙️ When objects are inserted or loaded from a file, you may need to align them before generating analysis input data. There are two ways to align objects. The first is to use the “Join Faces” function, and the second is to use the Boolean “Union” operation.

There is a very subtle but important difference between when using “Join Faces” versus “Union”. A union creates a single part, whereas “Join Faces” leaves the

original parts, but they are very close together. Separate objects can be assigned different materials and boundary conditions for an analysis application.

Join Faces Operation

 To join two objects using two faces:

- Activate Face Picking Mode (**p** **f** from the keyboard).
- Click on the face of the first object to be aligned. This will be the "from" object.
- Click on the "to" face that you wish to align with the "from" face.
- Press **j** or **J** ("Join"). The "j" case joins using the minimum angle between the faces, and "J" uses the maximum angle. The latter case can be used when you want to flip the object over.
- You will be prompted for the "Object Spacing", which in this case is the spacing between the faces. If this is set to 0.0 the faces will be aligned exactly. However, you may want to set this to the particle size that you will use later so that generated particles are not too close.
- Select **OK**. The "from" face" will be rotated and moved so that it aligns with the "to" face.
- Undo a Join by typing **Ctrl+Z** or from the **Edit** → **Undo** menu item.

Objects are aligned using normal vectors to the selected faces. There are two normal vectors to any face pointing in opposite directions. The meaning of "maximum angle" and minimum angle is ambiguous when rotating 90° or 180°, so you may have to try both "j" and "J" to see which works for the particle case.

Boolean Operations

 The following Boolean operations may be performed from the Tools menu:

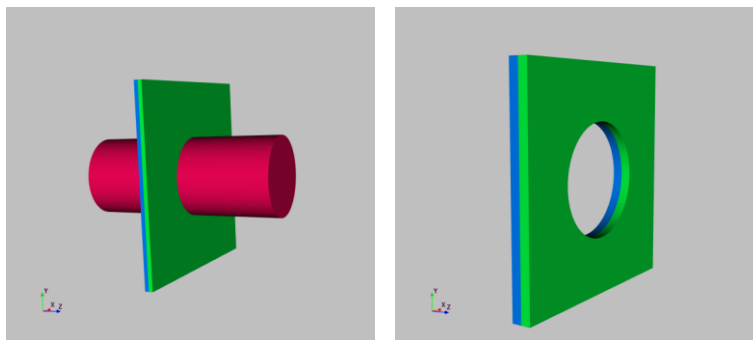
- Difference. The first object selected becomes a “cutter”. The intersecting areas of the cutter and each of the other selected objects is removed, and then the cutter is deleted. This operation is useful, for example, to create holes in other objects.
- Difference (cut only). Same as above, but the cutter is not deleted.
- Intersection. Create an object containing the common portion all selected objects. The most recently selected object is replaced.
- Union. Create a single object from all selected objects. The most recently selected object is replaced. See also [Join Faces](#).

Difference and Intersect operations may be applied to STEP and IGES data, and Union may be used with any type of data.

Boolean Operation Example

In the following example we wanted create two plates which in the analysis program will consist of different materials. Additionally, a hole is required through the plates. This data can be created as follows:

- Create two plates. Shift one plate with the Geometry Editor so that it doesn't overlap the other plate.
- Create a pipe with the inner radius set to zero.
- Select the pipe. The first object selected becomes the cutting tool.
- With the **[Shift]** key pressed, select both plates in either order.
- Use the **Boolean** → **Difference** operator to cut the pipe out of both plates. The pipe is deleted by the Difference operator.



Using the Difference operator on multiple objects is handy in this case—otherwise you would need to insert and the cutter multiple times.

Advanced Cut Operations


The Difference Boolean Operation can often be used to make simple cuts, but more complex cuts may require several steps. One particular case is described here.

Suppose you have a CAD object such as an engine block from which you want to create a die casting mold. Therefore, you need the engine block, a box to represent the mold, and a pipe going through the mold into which you will inject liquid metal. You can easily join a pipe to the engine block and then add a box, and you can then use the Difference operator so that the pipe will cut through the box.

However, the Difference operation deletes the cutting tool (the pipe), and once you have added the box it is difficult to add back a replacement pipe with the same alignment. The solution to this is:

- Instead of Difference, use Difference (cut only). This breaks up the CAD object into three parts: the mold minus the exterior of the pipe, the exterior of the pipe, and finally a disk representing the interior of the pipe (which we do not want).
- Select the mold, and apply the Tools → Split Compound operation. This splits the mold into two pieces: the remainder of the mold (minus the exterior of the pipe) and the disk.
- Select the disk and delete it.


Create Hollow Solid

 Some CAD objects such as in STEP files are represented only by boundary representation. If you try to export this as a Point Cloud you will end up with a solid block of particles. This is because the visible walls in the model actually have no thickness, so the conversion process thinks that the entire object is a single wall. However, in most cases you want the inside to be hollow, which you can achieve as follows:

- Select one of the major faces, such as the frontmost face (type to enable face picking, and then click the face.)
- Select the **Tools** → **Create Hollow Solid** menu item to display the Create Hollow Solid dialog.
- Fill in the “Wall Thickness” value. The units are the same as in the object unit.
- Select , and each face is “thickened” in the inward direction.

Now you can export the object as a Point Cloud with hollow insides.


Repair Shape

 The **Tools** → **Repair Shape** menu item performs basic repairs to objects such as connecting disconnected edges and combining edges that are very close together into a single edge.

You will be prompted for:


- Precision. The general precision limit used by the repair tool.
- Maximum Tolerance. The limit to use when determining if the objects such as edges are “close”.
- Minimum Tolerance. The limit to use when determining when objects such as vertices are close enough such that they can be combined.

Subdivide Faces

 The **Tools** → **Subdivide Faces** menu item splits all faces such that no face has an area larger than the specified size.

You will be prompted for the desired maximum area.

Subdivide Surfaces

 The **Tools** → **Subdivide Surfaces** menu item splits surfaces with extreme angles into multiple surfaces having the specified maximum angle.

You will be prompted for the maximum desired angle.

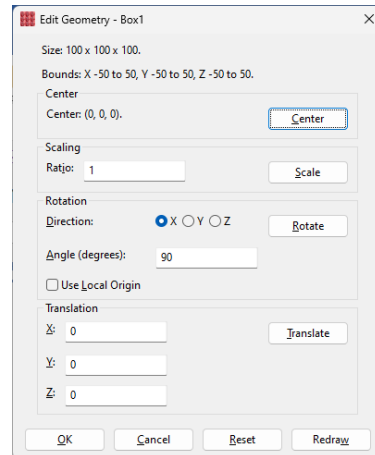
Edit Geometry

When you move or rotate objects with the mouse, you are actually moving the camera. However, when you are combining objects to be used for analysis, you may sometimes need to edit the underlying geometry itself using the Geometry Editor. Operation is as follows:

Keys: Ctrl+G: Edit Geometry.

Menu: Object → Edit Geometry.

- Select the object to be edited with a right-click. You may also type **n** (next) or **N** (previous) to cycle through and select objects.
- Open the Geometry Editor by pressing **Ctrl+G** or from the **Tools** → **Edit Geometry** menu item.
- Use the dialog settings to perform the desired operation(s):



- Center the object at (0, 0, 0).
- Scale (resize) the object.
- Rotate the object. Rotation is counterclockwise at the angle specified in degrees. If “Use Local Origin” is checked, rotation is around the center of the selected object, and if unchecked rotation is around the scene.
- Translate (move) the object.

Click the corresponding button to apply the transformation.

- Select **OK** to make apply the changes to the in-memory working data. The original on-disk copy is not modified unless you resave the object to a file.
- Undo a geometry edit sequence by typing **Ctrl+Z** or from the **Edit** → **Undo** menu item.

The Geometry Editor is intended for use with CAD data formats such as STEP and IGES. You may also edit Point Cloud data with this editor, although we are not sure if this is actually useful.

Hints

- To reset viewpoint after making extreme modifications, use Zoom In, Zoom Out, and the Camera menu and its shortcuts such as **Ctrl+F** (“View from front”).
- TenGun tries to avoid excessive redrawing while you’re editing the geometry. However, if the object goes out of view you can click the **Redraw** button to readjust the camera.


- The modified geometry's new position and orientation become the new values used by the  ("Home") button.

Image Display

TenGun can display BMP, JPEG (JPG), PNG, and some TIFF image files. This feature is provided mainly as a convenience so that you do not need to start a separate application to view images.

TenGun can also display DICOM (Digital Imaging and Communications in Medicine) images. Please note the following:

- DICOM files must have an extension of “.dcm”. Sometimes these files have no extension, and we are therefore expected to read the DICOMDIR directory file. However, we do not currently read this file at this time although may add this capability in the future as we collect more data.
- The “Next” and “Previous” keyboard commands will scroll through a series of DICOM images or through a multislice DICOM series.

Text Editor

TenGun includes a full-featured text editor. This is based on the popular “Scintilla” editor plugin from www.scintilla.org.

This editor is primarily a source code editor, and it therefore contains a number of features such as syntax highlighting. At the moment, these programmer-friendly features are not accessible from within TenGun, but we may enable them in the future.

With the extra source code handling features and internal storage of text as Unicode, the editor tends to be a bit memory-intensive. Therefore, you should not try to edit huge data files with this editor if you have a limited amount of memory. Additionally, even if you have a lot of memory, the maximum file size is currently 2GB.

This document does not discuss editor operation in detail. Please refer to the “[TenGun Quick Reference](#)” (available from the **Help** → **Quick Reference** menu item) for a table of keyboard commands.

Performance Hints

Performance on large datasets using the graphics capability built-in to the CPU is horrible. Therefore, if you have a GPU device you definitely want to set this as the preferred graphics device for this application.

For NVIDIA on Windows:

- Right-click on a blank area on the Desktop and select "NVIDIA Control Panel".
- Under "Manage 3D Settings" select "Program Settings".
- Use the Add function to add an entry for TenGun.exe. (Browse to where TenGun is installed to find the exe file.)
- Click "Apply" to activate the new preference.
- Close the NVIDIA Control Panel.

Experimental Features

Certain features in TenGun are marked as “experimental”. These are features that may not work perfectly either because something is beyond our control, or because the feature itself is borderline impossible. Therefore, features marked as “experimental” are excluded from our standard warranty. Included in this list are:

- The file browser. The basis of this feature is within a library that we use. It is known to sometimes fail with a yet-to-be determined complex sequence of mouse events. The same code also has difficulty dealing properly with removable drives, and has slow response when a slow network connection is involved in the view. The file browser is very handy when it works, but please keep these limitations in mind.
- Point Cloud Export, Interior Points. This feature does not work for all geometries, as discussed in this document.
- LS-DYNA support. We have only a limited number of LS-DYNA files, and we know that some cases do not work. So that we may remove the “experimental” status, any files that you can share with us would be greatly appreciated.