

# **WarpLAB**

VERSION 1.7

## **USER'S MANUAL**

Edited by Steve Maas

*Last updated: June 2, 2006*

# TABLE OF CONTENTS

<b>1. Getting Started</b>	<b>2</b>
1.1. Features of WarpLAB	2
1.2. Running WarpLAB	2
1.3. The Project File	3
1.4. Creating a New Project	3
1.5. Opening an existing project	4
<b>2. Tutorial</b>	<b>5</b>
2.1. STEP 1: Opening the sample project file	5
2.2. STEP 2: Setting some options that effect the display	6
2.3. STEP 3: Saving a screenshot	7
<b>3. The Environment</b>	<b>8</b>
3.1. The GUI	8
3.2. The Menu	8
3.3. The Toolbar	10
3.4. The Command Line	10
3.5. The Command Bar	11
<b>4. Post Processing</b>	<b>13</b>
4.1. Setting Post Processing Options	13
4.2. Summary View	15
4.3. Trackview	16
4.4. Making Plane Cuts	17
4.5. The Material Manager	18
4.6. Displaying user data	19
4.7. The View Settings	19
4.8. Saving Graphics	20
<b>5. Image Processing</b>	<b>21</b>
5.1. Setting Image Processing Settings	21
5.2. Image Histograms	21
5.3. Image Inverting	21
5.4. Volume Rendering	21
5.5. Exporting Image data	22
<b>APPENDIX A – Commands</b>	<b>23</b>
FILE COMMANDS	23
POST COMMANDS	23
TRANSFORM COMMANDS	24
EDIT COMMANDS	24
VIEW COMMANDS	24
HELP COMMANDS	25

# 1. Getting Started

## 1.1. Features of WarpLAB

WarpLAB is a finite element (FE) post-processing application that is specially designed to post-process *warping* problems. Besides common features found in many other FE post-process applications, such as generating surface plots, isosurface plots, plane cuts, and many more, it also has a unique set of features to analyze the images that are required to run a warping problem. A short list of features supported in the current version:

- Graphical User Interface (GUI)
- Command Line Interface
- Command Line Calculator
- Diverse plotting modes for visualization
- Image processing
- Volume rendering
- ...

The GUI of WarpLAB allows the user to interact with the FE model in an easy and intuitive manner. For the more advanced user WarpLAB also offers a command line alternative.

## 1.2. Running WarpLAB

WarpLAB runs on Windows XP, LINUX and Mac OSX. In order to run WarpLAB a memory size of at least 512 Mb is recommended. Some larger models might require even more memory. A processor of at least 1 GHz is also recommended. The computer's graphics card must support OpenGL, since the application makes extensive use of this 3D library.

### Command Options

When starting WarpLAB from the command line, some options can be given. The full syntax for the command is:

```
>warplab [file.wpp]
```

Where *file.wpp* is a WarpLAB project file (see section 1.3). If a filename is specified, the file will be loaded into the WarpLAB environment when WarpLAB executes.

In order to use the help files from within WarpLAB for the LINUX and Mac version, you are required to manually set the path to the WarpLAB main directory in the `.cshrc` (c-shell) configuration file. So, assuming that `/home/user/warplab/` is the WarpLAB's home directory, you'll add the following line to the `.cshrc` file:

```
setenv WARPLAB_HOME /home/user/warplab/
```

### 1.3. The Project File

In order to start a warping analysis, a project file needs to be created. The project file is a simple ASCII file that lists all information that WarpLAB needs to start the post processing. The project file can either be made by the user itself, or can be created automatically by WarpLAB by selecting the *File/New* menu (see 1.4. *Creating a new project*). The following is an example of a project file:

```
# WarpLAB project file sample
plotfile = n3plot
template = temp.raw
target   = study.raw
dimension = 64 64 8
range    = 0 0 0 9.35 9.35 7
```

**Box 1. An example of a project file.**

All lines beginning with a pound (#) symbol are comment lines. Next, the five basic “ingredients” for a warp project are defined.

```
plotfile    = base name of the plot file family
template    = filename of template image
target      = filename of target image
dimension   = image resolution in three dimension (nx, ny, nz)
range       = geometrical range of image data (xmin, ymin, zmin, xmax, ymax, zmax)
```

The lines can be placed in any order but all five fields must be present. In addition to these mandatory fields, a few optional fields can be given as well:

```
title       = Title of project. This title will replace the title in the plot file.
type        = Type of image. This specifies the type as an 8-bit or 16-bit image. If no
              image type is specified, it is assumed to be the 8-bit default type.
```

### 1.4. Creating a New Project

To create a new project, the user can create the project file itself, e.g. by using a word processor to write the ASCII file. However, WarpLAB can create the project file for you. If you select the *File/New* menu, a dialog box will show up in which the user can type in all the necessary information. After pressing “OK”, WarpLAB will create the project file with the name given by the user and automatically opens the project.

**Figure 1. The New Project dialog box**

Figure 1 shows the New Project dialog box. Enter the required information to create the project file. The *Project Name* is the name that will be given to the project. WarpLAB will attach to this name the extension *.wpp* and create the file for you with the information provided. *Plot base* is the base name for the family of plot files. This is usually the name of the first plot file outputted by NIKE3D, e.g. *n3plot*. *Template* is the name of the template image file. *Target* is the name of the study image file. *Image type* is the type of file image, specified as 8 or 16 bits, 8-bits being the default image type. *Dimensions* are the dimensions of the image data. Note that it is assumed that both template and target have the same dimensions. *Range* is the geometrical range (in the same units as the model's coordinates) of that image data. Note that at this point WarpLAB only supports 8-bit raw image data.

After the user selects the OK button, the file will be created and saved to disk in the current working directory. Next, WarpLAB will attempt to read the file in and load the project's data. Note that the project file will always be created, regardless of whether it can successfully be loaded.

### **1.5. Opening an existing project**

To open an existing project, select the *File/Open* menu. This will display the Open File dialog box. Select the project file (extension *\*.wpp*) and click on OK. The file will then be loaded into WarpLAB.

## 2. Tutorial

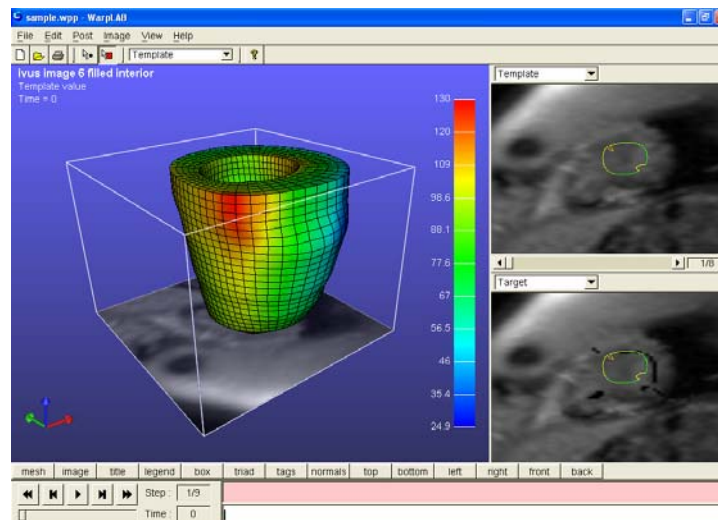
This chapter describes some of the main features of WarpLAB by using a sample file that comes with the executable. The sample shows a part of the left ventricle of the heart that was warped from beginning to end of diastole.

In this tutorial you will learn:

- to open an existing project
- position the model in the graphical view
- change a few post-processing options
- saving a screenshot of the display

### 2.1. STEP 1: Opening the sample project file

To open the sample project, just open the file `sample.wpp` as described in section 1.5. This should load the sample data into WarpLAB. After loading the display should look similar to figure 2.



**Figure 2. Screenshot of the sample project file.**

In the left view (the *graphics view*) you can see the FE model. The color display by default shows the current template values. In the upper-left corner, below the title of the project, you can see what is being displayed. In this case it will read the default value *Template value*. The vertical colored bar on the right of the graphics view shows the legend.

On the right you can see two image views. By default the upper view will show the (un-deformed) template image and the lower view the target image. You can use the slide bar located between the image views to scroll through the different images. Also note that the current slice is also displayed in the graphics view on the left.

In the lower-left corner, below the graphics view, the time controller is displayed which allows you to move through the different time steps of the FE analysis.

## 2.2. STEP 2: Setting some options that effect the display

**Moving to the last timestep.** In this step we will change a few settings so we can display what we really want. The first thing we'll do is go to the last timestep. This is done by selecting the far right button (>>) on the time controller. This will advance the solution to the last timestep. You can also push the *next timestep* button (>|) to advance one step at a time until you reach the final timestep. Still another way is, when positioned on the very first step, to push the *previous timestep* (<|) button which will loop the solution to the last step. To continuously loop through each timestep, the *play* (>) button can be pushed.

**Positioning the model.** The user can rotate the current view of the model by simply holding down the left mouse button while dragging the cursor over the graphics view. By click+dragging the right mouse button, the user can also zoom in or out. To move the model left, right, up or down the user needs to click+drag the middle mouse button. Now practice this and position the model so that you have a clear view of the inside of the model.

**Changing the surface expression.** By default, when loading a project, WarpLAB will display the template value on the surface. To change this, open the post options dialog by selecting the *Post/Options* menu. You should see the Options dialog box, such as displayed in figure 3.

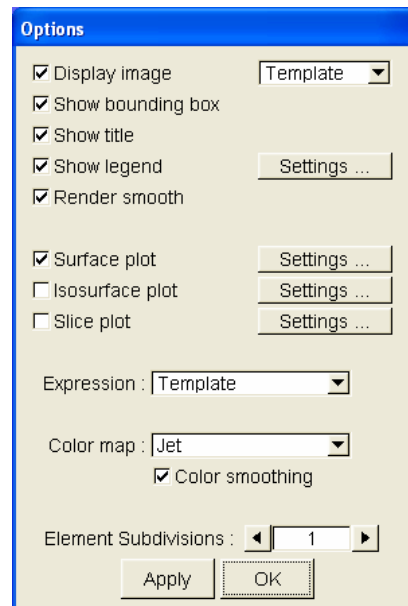


Figure 3. The Post-options dialog box.

To change the expression for the surface, select the drop-down box named *Expression*. Now select *Von Mises Stress*. Also, in this dialog box, unselect the *Display image* checkbox. This will remove the current image slice from the graphics view. After selecting OK, the graphics view will be updated in accordance with the new settings.

### **2.3. STEP 3: Saving a screenshot**

In this step we will save the current graphics view to file. To do that, select the *File/Print* menu. This will open the Save File dialog box. Enter the filename you wish to use (e.g. sample.tif) and press the OK button. WarpLAB by default saves files as TIFF files (extension .tif), but can also save bitmap files (extension .bmp). To save a file as a bitmap file, select *Bitmap Files* in the drop-down box, named *Show*.

After selecting OK, the current graphics view will be saved to a file, which can then be opened and modified by any image processor that supports the provided file types.

This completes our tutorial. The points to remember are:

- Use *File/Open* to open an existing warping project file (\*.wpp)
- You can use the mouse to position the model by left or right click+dragging
- The *Post/Options* menu shows a dialog box where you can change the post processing options
- The *File/Print* menu you can use to save the current graphical view to an image file

## 3. The Environment

### 3.1. The GUI

WarpLAB comes with a Graphical User Interface (GUI) that allows an easy and intuitive way to interact with the FE model and image data. Figure 4 shows the GUI when the program starts and after the sample file is loaded.

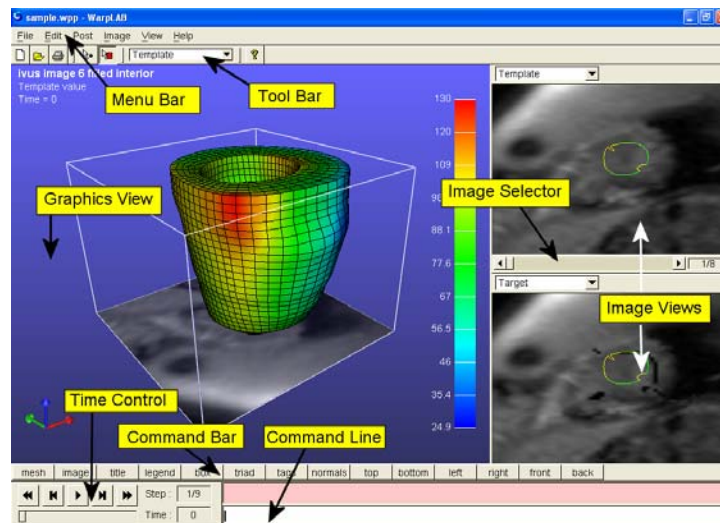


Figure 4. The GUI after the sample file has been loaded

The GUI consists of several items. The *Menu Bar* at the top (also see section 3.2) allows access to the various WarpLAB options and settings. Located below the menu bar is the *Tool Bar* (see section 3.3). The *Graphics View* (GV) shows a 3D image of the FE model. On the right of the GV are displayed two *Image Views* that allows the user to look at two different images simultaneously (by default these are the template and target images). A drop-down box, located above each of the image views, allows the user to select what image data he would like to see. The *Image Selector*, located between the image views, is used to scroll through the image data in the event that more than one image slice is present. In between the GV and the IV's, a small vertical bar is located which can be click+dragged to rescale the view areas. At the bottom left, the *Time Controller* is used to select the timestep that is being displayed. To the right of the *Time Controller*, is the *Command Line*, this can be used as an alternative to invoke WarpLAB commands. The *Command Bar* lists several buttons with common commands.

### 3.2. The Menu

In this section we will give an overview of the different menu items that can be found in WarpLAB. The main menu bar, located at the top of the GUI window, consists of the following items:

- *File* - file operations, such as opening a WPP (warp post-process) project
- *Edit* - options that apply to the current selection, such as hiding, un-hiding, etc
- *Post* - do some post-process operations and set the post-processing options
- *Image* - provides options and operations to be done on the image data

- *View* - provides options to customize the *Graphics View*
- *Help* - allows access to WarpLAB help and the About dialog box

A more in-depth description of the different menu items follows.

## The File Menu

The file menu gives access to file operations and has the following menu items:

- *New* - Starts a new warping post-process project
- *Open* - Opens a project file (\*.wpp)
- *Project Info* - Gives some information about the current model and project
- *Print* - Prints the current view to a file
- *Export image data* - Exports the various image data
- *Export Scene* - Export the data of the current scene to file
- *Exit* - Quits the application

## The Edit Menu

The edit menu gives access to operations that apply on the current selection:

- *Hide Selected* - hides the current selected elements
- *Hide Unselected* - hides the non-selected elements
- *Invert Selection* - inverts the current selection
- *Unhide All* - Un-hides all the previously hidden elements
- *Select All* - selects all visible elements (invisible elements remain unselected)
- *Clear Selection* - Clears the current selection

## The Post Menu

The post menu allows the user to do post-processing operations and also allows you to set many of the options that effect what is being displayed.

- *Options* - Set many of the options that effect what is being displayed
- *Summary* - Provides some 2D plots of several fields such as displacement, strain, stress, etc.
- *Plane Cut* - Displays a plane cut view of the model.
- *Trackview* - Shows a time history of selected nodes or elements
- *Material Manager* - Displays the *Material Manager*

## The Image Menu

The image menu, allows you to change the settings that effect the displaying of the images, as well as perform some image analyses.

- *Options* - Set options that effect image display
- *Histogram* - Shows histogram data for template and target
- *Invert* - Inverts the image data

- *Volume render* - Show a volumetric rendering of the image data

## The View Menu

The view menu allows the user to choose options that customize the display.

- *Settings* - Set foreground and background colors and style







## The Help Menu

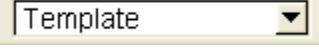
The help menu allows access to the help features available in WarpLAB and access to the software version and developers information.

- *WarpLAB Help* - Opens the Help window
- *About* - Displays the about box

### 3.3. The Toolbar

The *toolbar* is located below the menu bar and allows quick access to several menu items:

-  Creates a new project.
-  Opens an existing project.
-  Saves a screenshot to file.
-  Switches to node-selection-mode.
-  Switches to element-selection-mode.
-  Displays the WarpLAB Help window.

 Dropdown box allowing the user to select the current displayed expression.

### 3.4. The Command Line

The command line, located at the bottom of the screen, gives the user an alternative way to invoke WarpLAB commands. Most of the commands can be issued from here by just typing the name of the command. In addition, the command line can also be used as a simple calculator.

#### Using the Command Line

Commands can be entered on the command line by typing the command name, followed by the command arguments, if any. For example, the command to open a file is simply *open*, followed by the filename:

```
>open sample.wpp [enter]
```

To repeat the last command, just type an exclamation mark (!). So, for example, the following two commands will advance the timestep twice:

```
>next [enter]
>! [enter]
```

Help for a command is also available by preceding the command name by a question mark (?). For example, to find out what the *title* command does, type:

```
>? title [enter]
```

Note the space between '?' and 'title'.

Appendix A gives a detailed description of all available commands. Some commands do not have a menu alternative and thus can only be invoked from the command line.

## The Command Line Calculator

On the command line simple arithmetic expressions can be evaluated by first typing the equal sign (=), followed by the expression. So, for example the following command,

```
>= 5.2+3*(4.1-3.6) [enter]
```

will result in 6.7.

The Calculator recognizes the basic arithmetic operators (+, -, \*, /), the power operator (^), floating point representation (e.g. 3.1415) as well as exponential representation (1.03e2), parentheses, a set of predefined named constants,

```
pi = 3.14159265
e = 2.71828182
```

and a set of arithmetic functions:

```
cos   = cosine of argument
sin   = sine of argument
tan   = tangent of argument
ln    = natural logarithm of argument
log   = base 10 logarithm of argument
sqrt  = square root of argument
```

### 3.5. The Command Bar

Located below the Graphics View, the *command bar* gives access to several useful commands. By simply clicking on the button, the label of which contains the desired command, the command is executed. Most of these commands (e.g. mesh, image, title,

etc.) are toggle buttons, so clicking it once deactivates the option and by clicking it again, the option is reactivated.

## 4. Post Processing

### 4.1. Setting Post Processing Options

Most of WarpLAB's settings that affect what is displayed in the Graphics View can be set by invoking the Options dialog box. This can be done by selecting the *Post/Options* menu, pressing the F2 shortcut or by typing `options` on the command line. Next, the Post Options dialog box will be displayed and looks like figure 5.

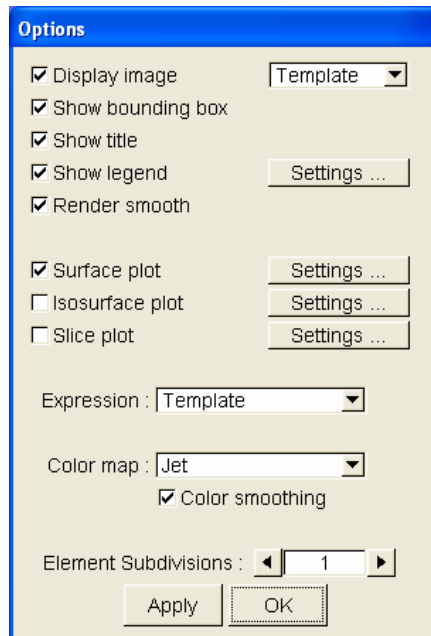


Figure 5. The Post Options dialog

The following settings can be set in the Post Options dialog box:

- **Display image:** show the current image slice also in the GV. In the drop down box, the user can select which image to display (template or target).
- **Show bounding box:** display a box that represents the geometrical range of the image data.
- **Show title:** display the title in the GV.
- **Show legend:** display the legend in the GV. By pushing the settings button next to it, additional parameters that affect the way the legend is displayed can be selected.
- **Render smooth:** if selected, uses Gouraud shading to display the plots. If not, flat shading is used.
- **Surface plot:** show a plot of the exterior surface of the model. The color of the model is based on the selected expression. Additional settings for this type of plot can be set by clicking the *Settings* button next to it.
- **Isosurface plot:** shows an isosurface plot of the expression. Additional settings for this type of plot can be set by clicking the *Settings* button next to it.

- **Slice plot:** shows a slice plot of the expression. Additional settings for this type of plot can be set by clicking the *Settings* button next to it.
- **Expression:** affects what is being evaluated and displayed as the texture on the mesh. (see Table 1 below for descriptions and corresponding commands using *eval command*).

Expression Field	Description	Command
<b>Image:</b>		
None	Displays no texture	none
Template	Displays template texture	template
Target	Displays target texture	target
Difference	Displays difference texture	diff
<b>Displacement:</b>		
X displacement	Displays texture displacement in the x direction	xdispl
Y displacement	Displays texture displacement in the y direction	ydispl
Z displacement	Displays texture displacement in the z direction	zdispl
XY displacement	Displays texture displacement as the xy shear	xydispl
YZ displacement	Displays texture displacement as the yz shear	yzdispl
XZ displacement	Displays texture displacement as the xz shear	xzdispl
Total displacement	Displays total texture displacement	displ
<b>Strain:</b>		
X strain	Displays texture strain in the x direction	xstrain
Y strain	Displays texture strain in the y direction	ystrain
Z strain	Displays texture strain in the z direction	zstrain
XY strain	Displays texture strain as the xy shear	xystrain
YZ strain	Displays texture strain as the yz shear	yzstrain
XZ strain	Displays texture strain as the xz shear	xzstrain
Effective strain	Displays effective stain of texture	vmstrain
Principle strain 1	Displays first principle strain component	e1
Principle strain 2	Displays second principle strain component	e2
Principle strain 3	Displays third principle strain component	e3
Prin dev strain 1	Displays first principle deviatoric strain component	e1dev
Prin dev strain 2	Displays second principle deviatoric strain component	e2dev
Prin dev strain 3	Displays third principle deviatoric strain component	e3dev
Max shear strain	Displays texture strain as the maximum shear	maxe
Plastic strain	Displays texture strain of plastic	plast
<b>Stress:</b>		
X stress	Displays texture stress in the x direction	xstress
Y stress	Displays texture stress in the y direction	ystress
Z stress	Displays texture stress in the z direction	zstress
XY stress	Displays texture stress as the xy shear	xystress
YZ stress	Displays texture stress as the yz shear	yzstress
XZ stress	Displays texture stress as the xz shear	xzstress
Von Mises stress	Displays texture as a Von Mises stress	vmstress
Principle stress 1	Displays first principle stress component	s1
Principle stress 2	Displays second principle stress component	s2
Principle stress 3	Displays third principle stress component	s3
Prin dev stress 1	Displays first principle deviatoric stress component	s1dev
Prin dev stress 2	Displays second principle deviatoric stress component	s2dev
Prin dev stress 3	Displays third principle deviatoric stress component	s3dev
Max shear stress	Displays texture stress as the maximum shear	maxs
Pressure stress	Displays texture stress resulting from pressure	press

**Table 1: Descriptions and commands of available expressions**

- **Color map:** selects the color map (Jet, Gray, Autumn, Red, etc.) that is used to color the mesh, based on the local values of the evaluated expression. The following are the color map values and corresponding commands: Jet=jet, Gray=gray, Autumn=autumn, Winter=winter, Spring=spring, Summer=summer, Red=red, Green=green, Blue=blue, Red-Black-Blue=rbb, Fire=fire.

- **Color smoothing:** If activated the colors are interpolated smoothly across the mesh. If not activated a discrete color map is used.
- **Element Subdivisions:** select the level of element subdivision. When drawing the mesh in the *Graphics View* the elements are divided in sub-elements to improve the visual appearance of the mesh. Note that this has no effect on Tet-meshes (Tetrahedral meshes).

Most of the options can also be set by invoking commands on the command line. See appendix A for a complete overview of the available commands.

## 4.2. Summary View

The Summary View displays an overview of minimum, maximum, and average values of different expressions. This summary of values can be calculated using only selected elements or nodes, or for the entire selection when no elements or nodes are selected. It can be opened by selecting the *Post/Summary* menu or by typing `summary` on the command line. Figure 6 shows the *Summary View* window.

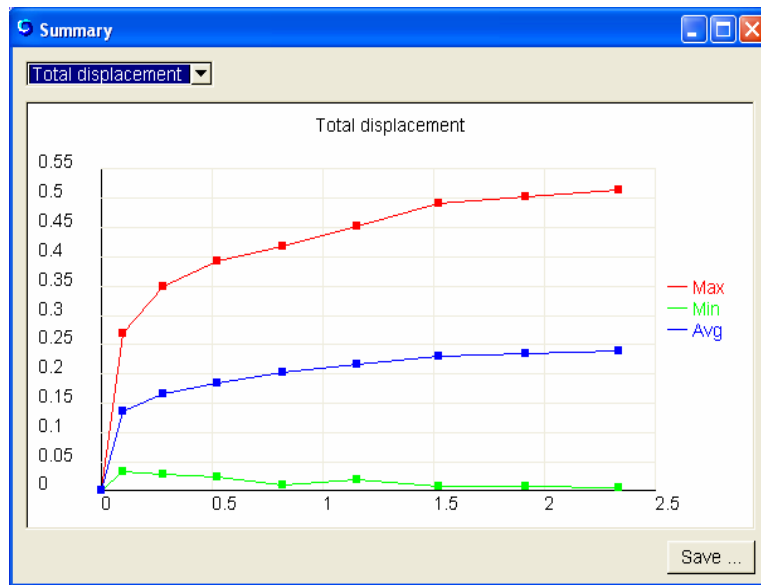


Figure 6. Summary View

To select the expression to display, click on the drop-down box in the upper left corner and select the desired expression. Each data point can be selected by clicking on it. The exact values will appear next to a selected data point.

To save the summary data to file, click the *Save* button located at the bottom of the *Summary View*. This will open the File Save dialog box. After the user enters a filename, the data is saved to file as a simple ASCII file.

### 4.3. Trackview

The Trackview can be used to show time history plots for individual elements or nodes. To display the trackview, select the *Post/Trackview* menu or type `track` on the command line. The Trackview should appear as in figure 7.

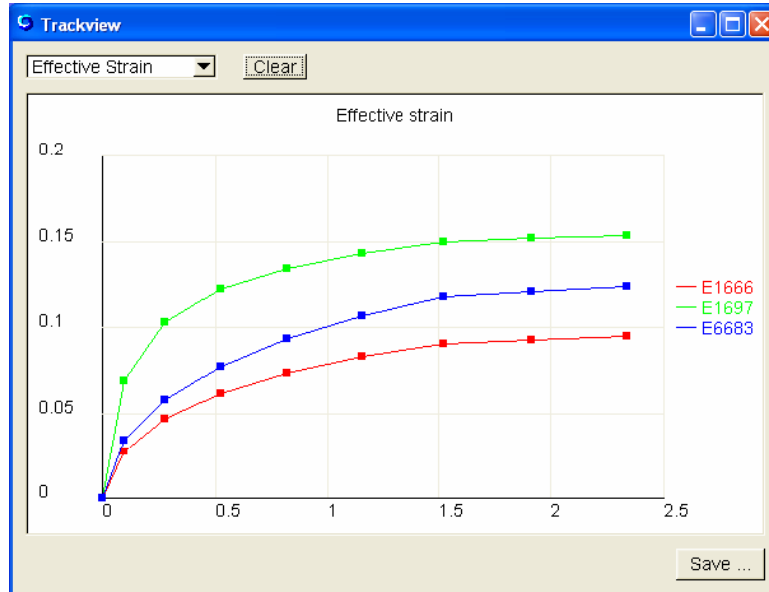


Figure 7. Trackview

The trackview displays the selected expression for the selected mesh items (see below on how to select mesh items). On the right the item numbers (preceded by an ‘E’ for ‘element’ or ‘N’ for ‘node’) are shown in the same color as the corresponding curve.

### Selecting mesh items

You can select both elements and nodes. Which item is selected is controlled by the two selection buttons in the *Toolbar*:



Switches to node-selection-mode.



Switches to element-selection-mode.

To add an item to the current selection, just shift+click the node or element. When tags are enabled, a dot followed by the item’s number will appear next to the item. To enable the tags, type `tags` on the *command line*, or select the corresponding button in the *command bar*. To remove an item from the current selection, just ctrl+click the node or element. You can (de-) select multiple items at the same time by dragging the mouse cursor while holding down the shift or ctrl key and the left mouse button. A colored rectangle will appear indicating what elements or nodes will be selected. Note that only *visible* elements or nodes that fall inside this rectangle will be selected. This means that only elements or nodes on the surface of the mesh can be selected.

You can also use the command line to select items. To select for example element 1000, just type the following command on the command line:

```
>selelem 1000 [enter]
```

This will have the same effect as shift+clicking the element (if it is visible). However, by using the command line, you can also select non-visible elements.

The command for selecting nodes is

```
>selnode 2500 [enter]
```

To clear the entire selection, you can either push the clear button in the Trackview dialog box (if it is opened) or you can use the command line by typing:

```
>clrsel [enter]
```

When no other windows are open (such as e.g. Trackview, etc.), pressing the ESC-key also clears the selection. Finally, you can select the *Edit/Clear Selection* menu to clear the entire selection too.

Also note that the *Edit* menu lists several options to manipulate the current selection, such as hiding, un-hiding, inverting, etc.

#### **4.4. Making Plane Cuts**

The menu *Post/Plane Cut* opens the Plane Cut window. This window shows the intersection of a plane and the model. The intersecting plane is displayed in the Graphics View. The position of this plane can be controlled by sliding the scroll bar at the bottom of the Plane Cut window. The orientation (in x, y, z) is controlled by sliding the scrollbars at the top. This more accurately controls the plane normal. The roller on the right side of the window controls the rotation of the plane around its normal and can be used to position the plane cut in a more desired orientation.

The color and expression used to display the plane cut are the same as in the Graphics View. In order to change them you need to change the expression and/or color map in the Post Options dialog box (see section 4.1).

You can also look at local values by clicking on a specific point on the plane cut view. A cross mark with the local value will appear on the selected point. This is a useful way to inspect the local values.

The *Print* button at the bottom of the *Plane Cut* window allows the user to save the current view to file.

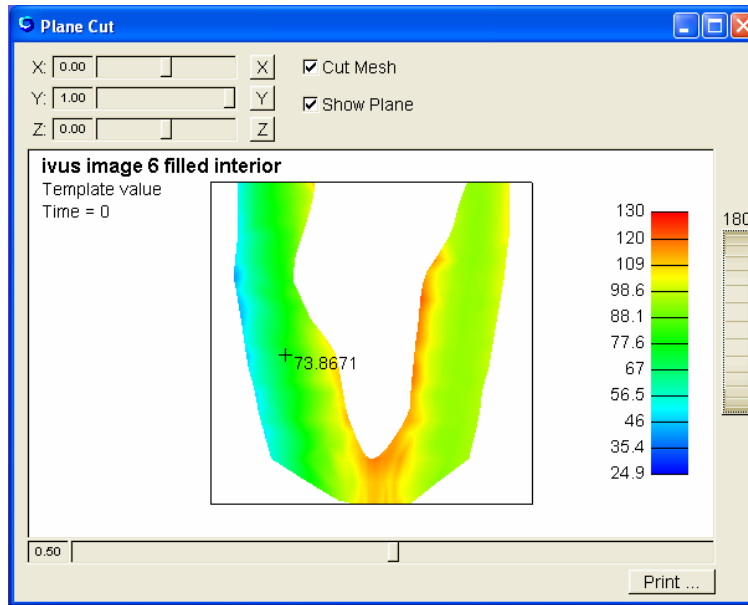


Figure 8. The Plane cut view. Clicking the mesh allows you to inspect local values.

#### 4.5. The Material Manager

The material manager controls the visual properties of the different materials in the FE model. It is opened by selecting *Material Manager* from the *Post* menu, typing `mat` on the command line, or by using the F3 shortcut. The Material Manager window is shown as figure 9.

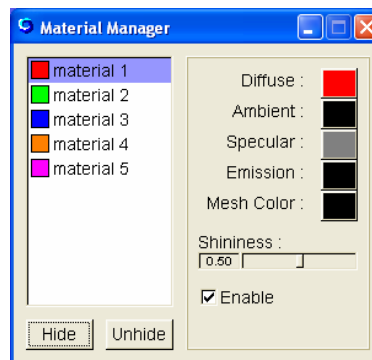


Figure 9. The Material Manager handles the appearance of the different materials.

In the left pane a list of all the different materials in the FE model is visible. The colored square, preceding the material name, identifies the diffuse color of the material. The current material is highlighted with a light blue background and can be changed by clicking with the mouse on one of the other materials.

The *hide* button hides the selected material and the *unhide* button un-hides it.

The right pane shows the settings that can be altered for the selected material. The options are:

- *Diffuse*: sets the diffuse color of the material. By clicking on the colored button, the Color Selector dialog is opened and the color can be changed. Closing the Color Selector will automatically update the color in the Graphics view. Note that the diffuse color is only used when no expression is selected or when the material is disabled.
- *Ambient*: sets the ambient color for the material. Click the colored button next to it to change the ambient color.
- *Specular*: sets the specular color for the material. Click the colored button next to it to change the specular color.
- *Emission*: sets the emission color for the material. Click the colored button next to it to change the emission color.
- *Mesh Color*: sets the mesh color for the material. Click the colored button next to it to change the mesh color.
- *Shininess*: sets the shininess value for the material. Slide the slide bar to change the shininess value.
- *Enable*: If checked the material is enabled and the color of the material is determined by the local expression value. If unchecked the material will be disabled and displayed with its diffuse color.

#### 4.6. Displaying user data

Besides displaying the expressions specified in the expression selector, the user can also display data that is imported from a file. In order to do this the user needs to use the `userdata` command. Which can load up to five different user data files. So, the following command

```
userdata1 "Deviatoric xy stress" dev.txt
```

will load the data in the file `dev.txt` into the first user data buffer and give it a description. This description will show up in the expression selector under the *user* menu and in the title on the graphics view.

#### 4.7. The View Settings

The *View Settings* dialog box (fig. 10 below) controls the background color, background style and foreground colors. You can invoke it by selecting the *View/Settings* menu or by typing `view` on the command line.

It allows you to select the following options:

- **Background color 1**: set the first background color. (The command line currently supports setting color to white, black, red, green, and blue).
- **Background color 2**: set the second background color. (The command line currently supports setting color to white, black, red, green, and blue).
- **Swap**: swaps the first and second background color.

- **Background style:** set the style of the background. The following are styles and corresponding commands: Color 1=color1, Color 2=color2, Fade Horizontal=horz, Fade Vertical=vert, Fade Diagonally=diag.
- **Foreground color:** sets the foreground color. That is the color of the title, the legend and the bounding box. (The command line currently supports setting color to white, black, red, green, and blue).

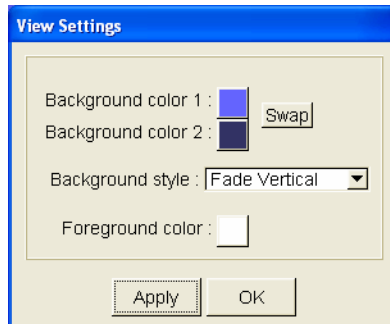


Figure 10. View Settings dialog box.

#### 4.8. Saving Graphics

To save a screenshot of the current Graphics View, select the *File/Print* menu. This will save the current view to a file. When the menu item is selected, the File Save dialog box appears and the desired filename can be entered or selected. Images can be saved as Windows Bitmap or TIFF images. In either case a 24 bit, uncompressed image is saved.

## 5. Image Processing

### 5.1. Setting Image Processing Settings

The settings that affect the way the image data is displayed in the image views can be set in the Image Options dialog box. This dialog box can be accessed through the *Image/Options* menu item. Figure 11 shows the Image Options dialog box.

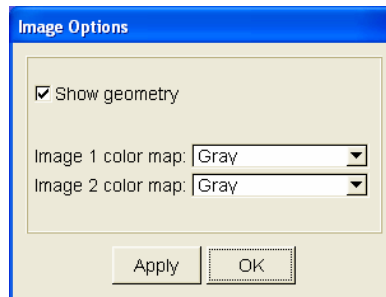


Figure 11. Image options dialog box

The **show geometry** checkbox is used to show or hide the current geometry in the image views. The geometry displayed is similar to the plane cut feature and shows the contours of the intersection between the geometry and the image plane.

The **Image 1 (2) color map** can be used to display the images with different color maps.

### 5.2. Image Histograms

The histogram of the template and target images can be inspected by selecting the *Image/Histogram* menu. The histogram of the template and target are displayed in different colors. By default the scale is linear, but a logarithmic scale can be used by checking the *Log* checkbox at the bottom of the dialog box.

### 5.3. Image Inverting

The image can be inverted by selecting the *Image/Invert* menu item. The resulting image will have a reversal of contrasting colors.

### 5.4. Volume Rendering

A volume rendering of the image data can be visualized by selecting the *Image/Volume render* menu item. A dialog will pop up that allows the user to set the parameters (figure 12).

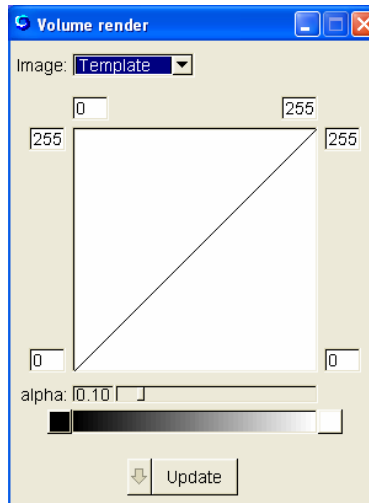


Figure 12. The Volume render dialog box.

In the Image field you can select to volume render either the template or the target image. The curve displays the linear mapping from the image grayscale values to transparency levels. So, in the case displayed in figure 12, black pixels (grayscale 0) will be completely transparent, while white pixels (grayscale 255) will be opaque. The edit fields surrounding the curve view allow you to customize the mapping. You can manually type in the values or you can click+drag in the edit fields to change their value. The *alpha* scale bar sets the global transparency level of the volume render. The color bar at the bottom shows the color mapping from grayscale to color. Pushing the buttons at both ends of the color bar will popup a color selection dialog.

The volume rendering is not updated until you push the *Update* button. To automatically update the volume rendering while you change the parameters you must set the *auto-update* option (i.e. push the button with the arrow, next to the *Update* button).

***WARNING! Volume rendering a 3D dataset is a computationally very intensive task. It is highly recommended to only use volume rendering if you have a graphics card that supports hardware acceleration. Also note that on most laptops graphics hardware acceleration is not activated when the laptop is running on batteries.***

## 5.5. Exporting Image data

The image data displayed in the Image Views can be exported by selecting the *File/Export Image data* menu item. This is particularly interesting for exporting the deformed template or difference images since they are generated by WarpLAB. However, since the images can be rescaled on export, it is also a useful tool to rescale the template or target image data.

The images are exported as 24 bit RAW images (in RGB colors). The images are 3D images and contain as many slices as the template (or target) data, but might have a different resolution. This resolution is displayed in the Export Image Data dialog box. In case the images depend on the deformation of the model (such as in the deformed template and difference images), only the images at the current timestep are exported.

## APPENDIX A – Commands

Here follows a list of all the commands recognized by WarpLAB. A command can have no arguments, optional arguments or mandatory arguments. The general syntax is:

*command* [*optional arg*] <*mandatory arg*> [*option A* |*option B*],

where *command* is the name of the command, [*a*] indicates optional arguments, <*a*> indicates mandatory arguments, [*a* | *b*] indicates a list of choices.

! repeats the last command

? <*command*> provides help on a *command*, where *command* is the name of the command.

= <*expression*> invokes the command line calculator.

### FILE COMMANDS

exit	Exits the application.
info	Displays information about the current model and project.
new	Opens the File New dialog box.
open [ <i>filename</i> ]	If <i>filename</i> is present, opens the file, otherwise opens the File Open dialog box.
print [ <i>filename</i> ]	Prints the current view to a file with name <i>filename</i> . If <i>filename</i> is omitted, the File Save dialog is opened.

### POST/IMAGE COMMANDS

box [ <i>on/off</i> ]	Shows (on) or hides (off) the bounding box in the Graphics View.
color < <i>colormap</i> >	Sets the color map. Refer to section 4.1 for valid <i>colormap</i> commands.
cut	Displays the plane cut view.
histo	Shows the image histogram window.
image [ <i>on/off</i> ]	Shows (on) or hides (off) the image in the Graphics View.
range [ <i>static/dynamic/user</i> < <i>min max</i> >]	Sets the range for the legend.
legend [ <i>on/off</i> ]	Shows (on) or hides (off) the legend in the Graphics View.
mat	Shows the material manager.
mesh [ <i>on/off</i> ]	Shows (on) or hides (off) the mesh.
options	Displays the post-options dialog box.
smooth [ <i>on/off</i> ]	Renders the model in smooth (on) or flat (off) shade.
summary	Displays the post-summary.
title [ <i>on/off</i> ]	Shows (on) or hides (off) the title in the Graphics View.
track	Displays the track view
triad [ <i>on/off</i> ]	Shows (on) or hides (off) the triad.
userdata( <i>n</i> ) <" <i>description</i> "> [ <i>filename</i> ]	Loads user data from file.

## TIME CONTROL COMMANDS

anim [fps]	Starts the animation, with optional framerate fps
first	Go to the first time step.
last	Go to the last time step.
next	Go to the next time step.
prev	Go to the previous time step.
stop	Stop the animation
time <time>	Go to <i>time</i> timestep, where <i>time</i> is a number between 1 and the max numbers of timesteps.

## FIELD SELECT COMMANDS

eval <expression>	Evaluates the expression. Refer to Table 1 of section 4.1 for valid <i>expression</i> commands.
-------------------	---

## EDIT/SELECTION COMMANDS

clrsel	Clears the selection.
hide	Hides the current selection.
selelem <elem id>	Selects element with element id <i>elem id</i> .
selnode <node id>	Selects node with node id <i>node id</i> .
tags [on/off]	Shows (on) or hides (off) the selected item's tags.

## TRANSFORM COMMANDS

back	Display back view.
bottom	Display bottom view.
front	Display front view.
left	Display left view.
right	Display right view.
pan <x> <y>	Sets the panning coordinates (x, y)
quat <x> <y> <z> <w>	Sets the orientation quaternion $q(x,y,z,w)$
rotx <angle>	Rotate angle degrees around the x-axis.
roty <angle>	Rotate angle degrees around the y-axis.
rotz <angle>	Rotate angle degrees around the z-axis.
top	Display top view.
zoom <z>	Sets the zooming coordinate (z)

## VIEW COMMANDS

bg [color name  r g b]	Set background color 1 either by name or RGB value (0-1).
bg1 [color name  r g b]	Set background color 1 either by name or RGB value (0-1).
bg2 [color name  r g b]	Set background 2 color either by name or RGB value (0-1).
bgstyle <style>	Set the background fading style. Refer to section 4.7 for valid style commands.
fg [color name  r g b]	Set foreground color either by name or RGB value (0-1).
view	Options the view settings dialog box.

**HELP COMMANDS**

about	Displays the about box.
help	Displays the WarpLAB help window.