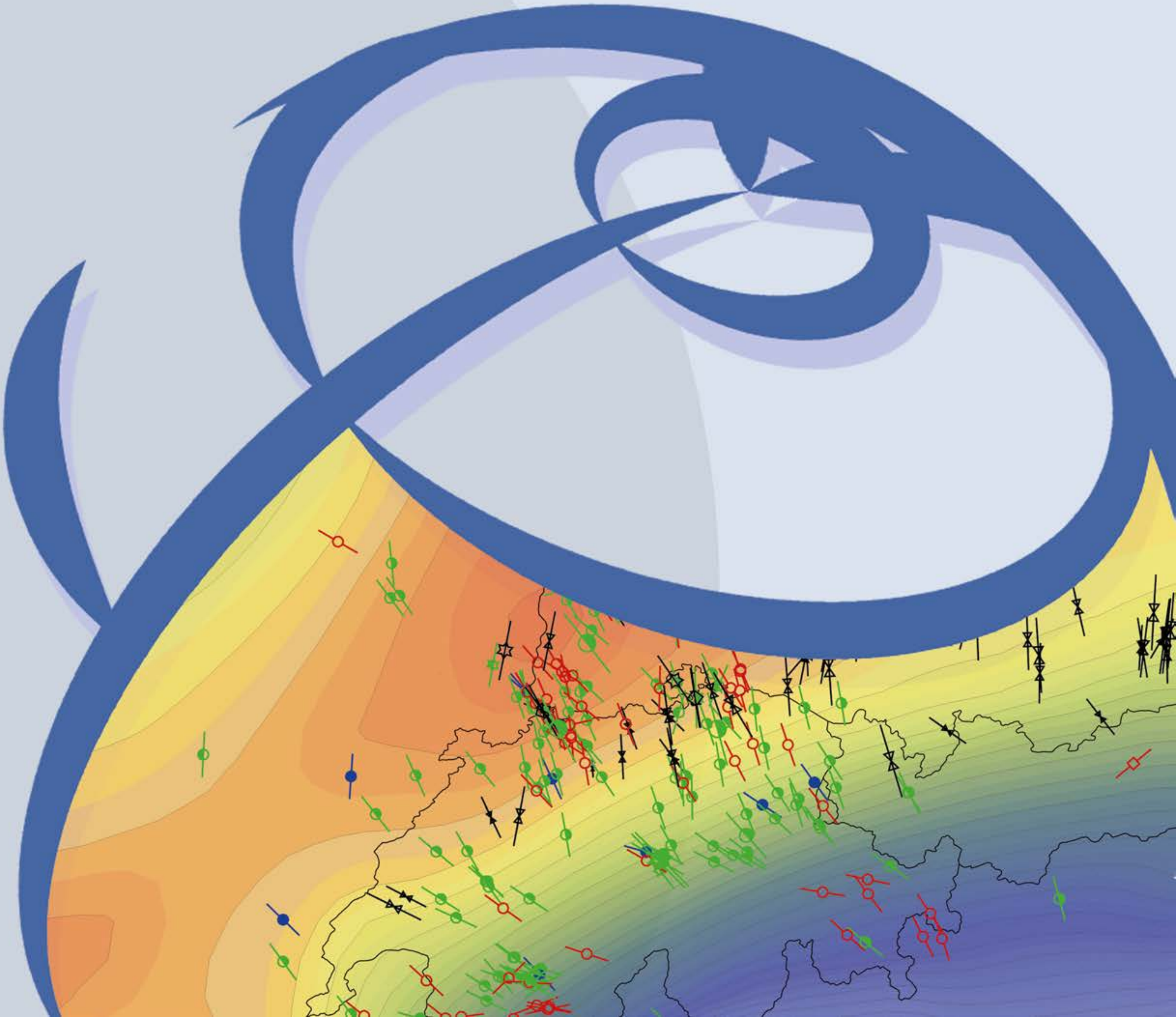
WSM WORLD STRESS MAP

WSM *Technical Report 20-01*

Manual of the Tecplot 360 Add-on *GeoStress v2.0*

Oliver Heidbach, Moritz O. Ziegler and Dietrich Stromeyer



Recommended citation of the report:

Heidbach, O., Ziegler, M.O., Stromeyer, D. (2020): Manual of the Tecplot 360 Add-on *GeoStress* v2.0. World Stress Map Technical Report 20-01, GFZ German Research Centre for Geosciences.
DOI: https://doi.org/10.2312/wsm.2020.001

The software described in this report, including data examples, is published as:

Stromeyer, D., Heidbach, O., Ziegler, M.O. (2020): Tecplot 360 Add-on *GeoStress* v2.0, GFZ Data Services.
DOI: https://doi.org/10.5880/wsm.2020.001

Previous version of the report:

Stromeyer, D., & Heidbach, O. (2017). Manual of the Tecplot 360 Add-on GeoStress (p.). GFZ German Research Centre for Geosciences. DOI: https://doi.org/10.2312/wsm.2017.001

A previous version of the software is published as:

Stromeyer, D.; Heidbach, O. (2017): Tecplot 360 Add-on GeoStress. GFZ Data Services.
DOI: https://doi.org/10.5880/wsm.2017.001

Imprint

World Stress Map Project
GFZ German Research Centre for Geosciences

Telegrafenberg
D-14473 Potsdam
Published in Potsdam, Germany
August 2020
https://doi.org/10.2312/wsm.2020.001

WSM Technical Report 20-01

Manual of the Tecplot 360 Add-on *GeoStress v2.0*

Oliver Heidbach, Moritz O. Ziegler and Dietrich Stromeyer
GFZ German Research Centre for Geosciences, Potsdam, Germany

Table of Contents

List of Tables

List of Figures

Abstract

For the visualization and analysis of the stress field from 3D thermo-hydro-mechanical (THM) numerical model results two main technical steps are necessary. First, one has to derive from the six independent components of the 3D stress tensor scalar and vector values such as the orientation and magnitude of the maximum and minimum horizontal stress, stress ratios, or the differential stress. It is also of great interest to display e.g. the normal and shear stress with respect to an arbitrarily given surface. Second, an appropriate geometry should be given such as cross sections, profile e.g. for borehole pathways or surfaces on which the model results and further derived values are interpolated. This includes also the three field variables temperature, pore pressure and the displacement vector.

To facilitate and automate these steps the Add-on *GeoStress* for the professional visualization software Tecplot 360 EX has been programmed. Besides the aforementioned values derived from the stress tensor the tool also allows to calculate the values of Coulomb Failure Stress (CFS), Slip and Dilation tendency (ST and DT) and Fracture Potential (FP). *GeoStress* also estimates kinematic variables such as horizontal slip, dip slip, rake vector of faults that are implemented as contact surfaces in the geomechanical-numerical model as well as the true vertical depth (TVD). Furthermore, the Add-on can import surface and polyline geometries and interpolates on these all available stress parameter.

This technical report describes the visualization tool with examples using 3D geomechanical-numerical model results from the finite element software Abaqus v2019. It also presents a number of special features of Tecplot 360 EX in combination with *GeoStress* that allow a professional and efficient analysis. We also address now the usage *GeoStress* with PyTecplot which is a powerful tool to automize the analysis. The Add-on as well as the example and input files used in this manual is published by Stromeyer et al. (2020) and the table below gives an overview of the files with a short explanation as they appear in the manual.

File Name	Explanation	Page
`example.plt`	Binary Tecplot format where the Abaqus results are stored	8
`example.lay`	Layout file that is controlling how the *.plt file is visualized	8
`example.odb`	Abaqus output file with model results that is loaded into Tecplot 360	9
`3D_View_Z_Displacement.sty`	Definition of a certain view style on a model result	16
`macro_example_RSR.mcr` `macro_example_RSR_edited.mcr`	Marco files that contain a number of Tecplot commands	17
`delta_K.eqn` `displacements.eqn`	Pre-defined mathematical operations that can be loaded in order to derive from the existing variables new ones	18
`GeoStress.dll`	Add-on *GeoStress* compiled for Windows 64 bit operating systems	21
`libGeoStress.so`	Add-on *GeoStress* compiled for Linux operating systems	21
`tecplot.add`	Edited Tecplot file where additional libraries have to be stated	21
`GeoStressCmd.dll`	The library compiled for Windows 64 bit operating systems	25
`libGeoStressCmd.so`	The library compiled for Linux 64 bit operating systems	25
`topograhy_nodes_only.txt`	ASCII input file of the model surface nodes in order to calculated the value True Vertical Depth (TVD)	31
`FP_plastic_properties.txt`	ASCII input file to calculate the Value Fracture Potential (FP)	38
`fault_1_N.txt, fault_1_S.txt` `fault_2_N.txt, fault_2_S.txt`	ASCII input file with nodes and elements that belong to a contact pair e.g. to calculate the relative displacement	55
`external_surface.txt`	ASCII input file with nodes and elements of an external arbitrary surface on which model results are mapped	59
`external_polyline.txt`	ASCII input file with nodes of an external arbitrary polyline on which model results are mapped	61

1 Introduction

1.1 Motivation and objectives

With the development of high-resolution 4D geomechanical numerical models that simulate the 3D stress tensor and its spatio-temporal changes due to natural (e.g. plate tectonics, earthquakes, erosion, sedimentation) and man-made processes (e.g. fluid injection or production, heat extraction, mining) the efficient and user-friendly visualisation of the model results became the most time consuming step within the model workflow. Typically, there are three issues that arise during the interpretation of the model results, i.e. in particular the stress tensor analysis:

1. The visualization capabilities provided in the post-processing tools of the commercial finite element packages do not provide scalar and vector values such as the tectonic stress regime, the orientation of maximum horizontal stress or stress ratios required in geosciences. Furthermore, the derivation of normal and shear stress with respect to arbitrary surfaces is not possible without major effort and internal programming in the post-processing codes.
2. The output figures are mainly based on screen shots. Thus, the combination with other data is limited and the quality of the figures that are produced by the software is mostly not acceptable for publication purposes. Furthermore, the access of the values in ASCII files for further post processing, publication in data repositories, and/or combination with other data (e.g. from a cross section through the values including the coordinates) is either not comfortable or even not possible.
3. The internal interpolation schemes are hardly documented, often inefficient and limited when it comes to the analysis of e.g. relative displacement along pre-existing faults in a model (simulated by so-called contact surfaces).

To significantly speed up the analysis as well as to increase the quality and flexibility of the model result analysis *GeoStress* has been programmed as an Add-on to the commercial visualization software Tecplot 360 EX (in the following we always refer to the version 2019 R1). Also, the Add-on *GeoStress* is independent from the incorporated finite element solver. Tecplot 360 EX is dedicated for the analysis and visualization of unstructured data that come from commercial finite element software packages, fluid dynamic codes and other numerical solvers. The data loader of Tecplot 360 EX can read the binary files from the leading commercial finite element codes (e.g. Abaqus or Ansys) and provides a reader for ASCII data. Tecplot 360 EX allows a professional programming of Add-ons in C or Fortran 90 and access to all model variables, details of the mesh and the incorporated interpolation schemes. The output can be ASCII, images in user-defined resolution postscript, vector files and movies in different output formats. The Add-on *GeoStress* is explicitly made for the analysis of the stress tensor and displacement vector, but Tecplot 360 EX can also handle the results of thermo-hydro-mechanical (THM) models and visualize the temperature and pore pressure field.

After this introduction, the report is followed by three parts. Chapter 2 provides a brief introduction into the basic functionality of Tecplot 360 EX with an emphasis on the key terms that are used in the present manual of the Add-on *GeoStress* and the accompanying application examples. For further details, we refer to the manuals of Tecplot 360 EX. In Chapter 3 we describe the installation of the Add-on *GeoStress* as well as its basic requirements and limits. Chapter 4 presents the functionality and usage of *GeoStress*, explains the physical and mathematical background and gives application examples from a 3D geomechanical-numerical model of the potential siting area for high-level radioactive waste Nördlich Lägern (NL) in Switzerland. Further details of the NL model setup are presented in the NAGRA report NAB 13-88 (Heidbach et al., 2013).

The report does not provide information on the geomechanical-numerical modelling workflow or the basic of the meshing of a volume into finite elements and the numbering of nodes and elements. For details on these issues we refer to publications such as Reiter and Heidbach (2014), Hergert et al. (2015) or Ziegler et al. (2016).

1.2 Changes with respect to *GeoStress* v1.0

Even though the general functions and the graphical user interfaces of *GeoStress* have not changed at all the technical changes in the background are significant. The three main changes of the new version 2.0 are:

1. Comprehensive re-organization of the *GeoStress* interpolation scheme for external surfaces and polylines. In *GeoStress* v1.0 all stress variables are derived from the 3D stress tensor components and then were interpolated on external surfaces and polylines. However, when these variables are bi-polar data, such as the S_{Hmax} orientation, the interpolation could have produced wrong results when the orientation was 179°N on one side of the surface and 1°N on the other side. The mean would have been 90°N which is obviously not correct. In *GeoStress* v2.0 all stress values on external surfaces and polylines are newly derived from the six independent components of the stress tensor. The only performed interpolation is that of the six independent components of the 3D stress tensor on the respective feature.
2. Version 2.0 of the Add-on *GeoStress* is compatible with the Tecplot 360 EX 2019 R1 version which was released in October 2019. This version can load *.odb files from the Abaqus version 2019. Furthermore, the Tecplot 360 EX 2019 R1 version is significantly faster in loading and processing model results. Furthermore, we now also provide, besides the 64-bit Windows version, a Linux-Version of the Add-on.
3. An additional library to use the functionalities of *GeoStress* together with Tecplot Macros or PyTecplot. This is of particular interest when large numbers of models have to be analysed automatically without starting the graphical user interface of *GeoStress*.

We also fixed a few minor bugs in the *GeoStress* v1.0 manual and polished a few issues on the user interface of *GeoStress*. Furthermore, we extended the manual with brief descriptions and examples how to use the library *GeoStressCmd.dll* for Windows with PyTecplot. We also provide with *libGeoStressCmd.so* a version for Linux operating systems.

1.3 Stress term definitions

In geoscientific and rock engineering literature, there is no agreement on stress term definitions. Thus, a brief description is given in Table 1.3-1 on how stress terms are used in this report.

Tab. 1.3-1: Definition of stress terms used in this manual.

Note that there is no strict definition of stress terms in the text books (e.g. Engelder, 1992; 1994; Brady and Brown, 2004; Jaeger et al., 2007; Zoback 2010; Schmitt et al., 2012).

Term	**Symbol**	**Definition/Comment**
In situ stress state	-	Undisturbed natural (virgin) stress state. In particular, the in situ stress is the sum of all natural stress contributions and natural processes that influence rock stress state at a given point.
disturbed stress state	-	Denotes that the in-situ stress is disturbed due to man-made changes such as impoundment, drilling, tunnelling, mining, fluid stimulation, reservoir depletion, or (re-)injection of waste water to name a few.
principal stresses	S_1, S_2, S_3	The symmetric stress tensor can always be transformed into a principal axes system (Jaeger et al., 2007). In the resulting diagonalized form the three remaining values are the eigenvalues and the principal stress directions correspond to the eigenvectors of the stress tensor. Per definition S_1 is the largest and S_3 the smallest principal stress ($S_1 \geq S_2 \geq S_3$).
differential stress	$S_d = S_1 - S_3$	Difference between the largest and the smallest principal stress.
mean stress	S_m	This is the first invariant of the stress tensor which is defined to be one third it's trace, i.e. $S_m = ⅓(S_{XX}+S_{YY}+S_{ZZ}) = ⅓ (S_1+S_2+S_3)$
effective stresses	S‘	Effective stresses S‘ are total stresses S minus pore fluid pressure p_f.
vertical stress	S_V	The magnitude of S_V is the integral of the overburden load. At greater depth the S_V orienation can deviate from the principal stress orientation.
magnitude and orientation of maximum and minimum horizontal stress	S_{Hmax}, S_{hmin}, S_{Hazi}	S_{Hmax} und S_{hmin} are the principal stresses of the 2D horizontal part of the 3D stress tensor with $S_{Hmax} \geq S_{hmin}$. Their orientation is uniquely defined by the angle clockwise from north. Assuming that S_V is a principal stress S_{Hmax} and S_{hmin} are principal stresses as well. S_{Hazi} is the orientation of S_{Hmax} in the 2D horizontal.
reduced stress tensor	-	It is often assumed that also at depth the orientation of S_V is a principal stress orienation. In this case only four stress tensor components are needed to describe the 3D in situ stress state: The S_{Hmax} orienation and the magnitudes of S_V, S_{Hmax} and S_{hmin}.
stress regime	-	Relates to stresses: The stress regime is an expression of the relative magnitudes of the principal stresses. It can be expressed as a continuous value using the Regime Stress Ratio (RSR) with values between 0 and 3 (Simpson, 1997). For further details see section 3.2.
tectonic stresses	-	According to Engelder (1992) tectonic stresses are the horizontal components of the in-situ stress state that deviate from a given reference stress state such as the uniaxial or the lithostatic stress state. In particular, these reference stress states imply that the magnitudes of S_{Hmax} and S_{hmin} are equal. However, definitions in the literature are not consistent or clear. Tectonic stress is not necessarily equal to the deviatoric stress which is the non-isotropic part of the stress tensor (the isotropic part is the mean stress S_m). This would be only true when S_V is a principal stress and assuming that the reference stress state is lithostatic i.e. the magnitudes of the three principal stresses are equal ($S_{Hmax}=S_{hmin}=S_V$).

Term	Symbol	Definition/Comment
tectonic regime	-	Relates to fault kinematics: Thrust faulting, normal faulting and strike-slip after Anderson (1905). Only when faults are optimally oriented in the stress field the stress regime is coincident with the tectonic regime (Célérier, 1995). Normal faulting: $S_V > S_{Hmax} > S_{hmin}$ Strike-slip: $S_{Hmax} > S_V > S_{hmin}$ Thrust faulting: $S_{Hmax} > S_{hmin} > S_V$

2 Basics of data visualisation with Tecplot 360 EX

2.1 Introduction

The following explanations are accompanied by an example using a geomechanical-numerical model of a potential siting area for high-level radioactive waste in Northern Switzerland (Heidbach et al., 2013). The model has several lithological layers with different elastic rock properties and the model volume is cut by two faults that strike approximately E-W (Fig. 2.1-1). After loading the initial equilibrated stress conditions due to gravity, horizontal displacement boundary conditions are applied to add the tectonic stresses. The model is shortened in N-W direction by 12 m and stretched in E-W direction by 1 m at each side. The Abaqus model result file (*.odb) contains four increments in one Abaqus step in which the displacement boundary conditions are step-wise increased. Since the model is purely elastic with Coulomb friction at the implemented faults, the increments are not time steps. Thus, the *Solution Time* given as 0.25, 0.5, 0.75 and 1.0 are arbitrary dimensionless numbers. E.g. in increment 0.25 the model is shortened by 3 m in N-S direction and pulled at the eastern and western boundary by 0.25 m each, respectively.

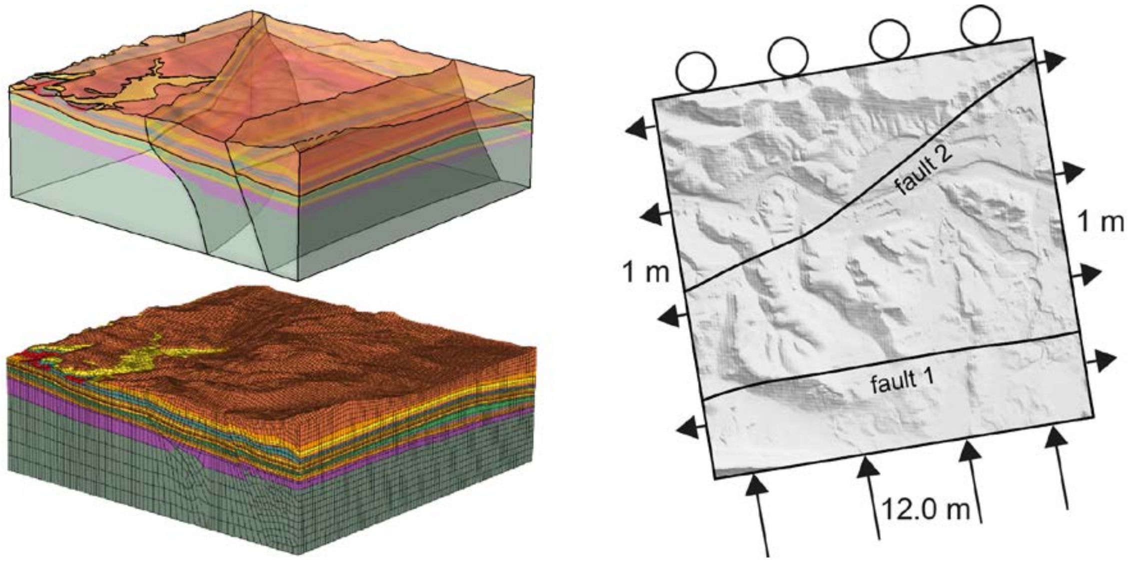


Fig. 2.1-1: Geometry, mesh and boundary conditions of the example model.

Upper left figure shows the lithological layering and the fault location that cut the model into three blocks. Lower left figure shows the model discretization into finite elements. The figure to the right shows the total displacement boundary conditions applied at the southern, western and eastern model boundaries. The circles on the northern boundary denote that no displacement is allowed perpendicular to the model boundary.

Further technical details of the model setup and interpretation of the results can be found in the report NAB 12-88 from Heidbach et al. (2013). The Abaqus results are provided in the file *example.odb*. The Tecplot-files are named accordingly *example.lay* and *example.plt*.

2.2 Basic concept and terminology

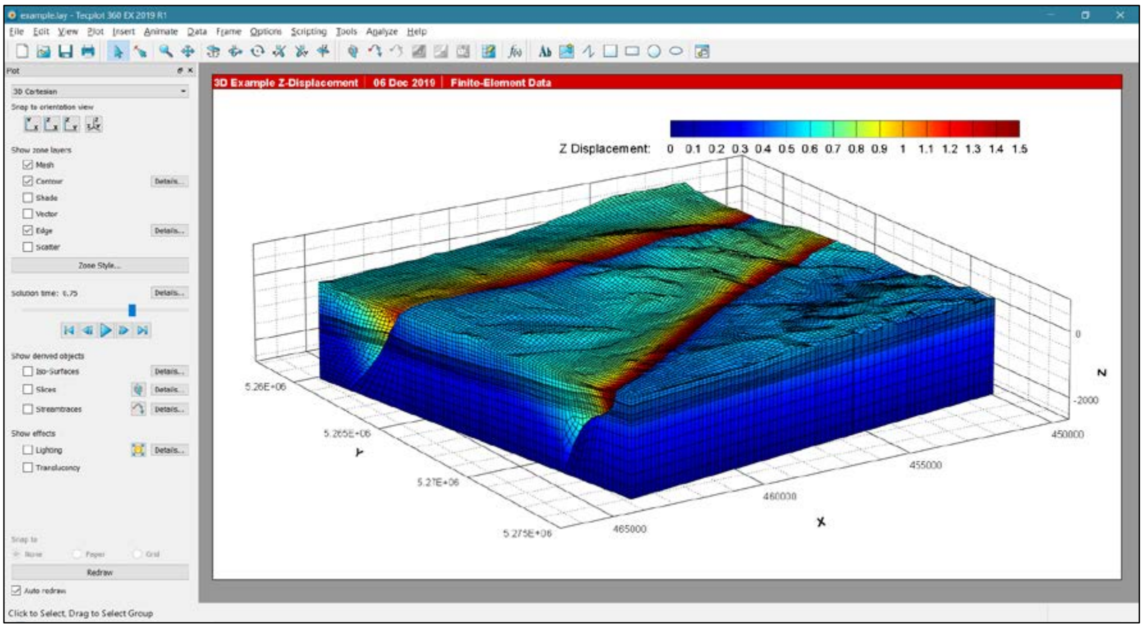
This section gives a brief introduction for the Tecplot 360 EX key words *Zone*, *Set of Zones*, *Time Strand,* and *Solution Time*. These terms are essential for the understanding how Tecplot 360 EX organizes the visualization of the model results. For further information, we recommend to read the Tecplot 360 EX manuals.

The graphical user interface of Tecplot 360 EX has four main areas (Fig. 2.2-1). The menu bar on the top offers all functions. Below the menu bar the tool bar offers several buttons for fast access to the key functions of Tecplot 360 EX. In the central large window Tecplot 360 EX displays the results of the selected *Zone* of your current analysis in frames. Different views or time steps of the model results can be displayed in several frames. The side bar on the left provides an easy access to the most common plot controls. The functions in the side bar depend on the plot type of the active frame. For 2D or 3D Cartesian plot types the *Zone Style dialog box* displayed in Fig. 2.2-2 can be used to select the *Zone*s that are shown in the active frame. For XY and Polar line plots the graphic is controlled by the mapping layer dialog box.

In general, the whole model data set is structured in Tecplot 360 EX in two levels: *Zones* and *Set of Zones*. A *Zone* can be an increment of an output file but also a later generated feature, such as a cross section. Each zone can be assigned to a *Solutiontime* to determine at which step in Tecplot the Zone is displayed. Zones are grouped together to *Set of Zones*. A *Set of Zones* is indicated by the variable *Time Strand*. Zones from different *Set of Zones*, i.e. with different assigned *time strands*, but the same S*olutiontime* are displayed simultaneously. An assigned *Time Strand* of 0 leads to displaying the zone during all *Solutiontimes*.

Fig. 2.2-3 and Fig. 2.2-4 show how cross sections (slices) over time are extracted. For this procedure enable the *Slices* option from the side bar (turn off *Contour*, *Mesh* and *Edge* in the side bar). Then choose from the dialog box the position of the slice that you would like to have. In order to interpolate the values from the volume onto the slice press the button *Extract slices….* on the bottom of the dialog box. In the upcoming dialog (Fig. 2.2-3) box you can set further details on the extraction. To actually see the four newly created *Zones* with a new *Time Strand* number the user can open *Data/Edit Time Strands…* from the menu bar and change the time strands of every newly created *Zone* therein. The new *Set of Zones* contains all *Zones* of the model. Again, each chronological and geometrical entity in each step of the Abaqus analysis is one *Zone*.

Note that with this procedure all values that exist in the model volume are interpolated by Tecplot 360 EX onto the slice regardless the potential errors that might occur when bi-polar data (e.g. S_{Hmax} orientation) are in the data set (see subsection 1.1 and 3.6 for further explanation). To avoid this you can use the *GeoStress* external surface function (subsection 3.7). With this function all values are calculated based on the six stress tensor components that are interpolated onto the surface.



3D Cartesian

Show zone layers
☑ Mesh
☑ Contour
☐ Shade
☐ Vector
☑ Edge
☐ Scatter

Solution time: 0.75

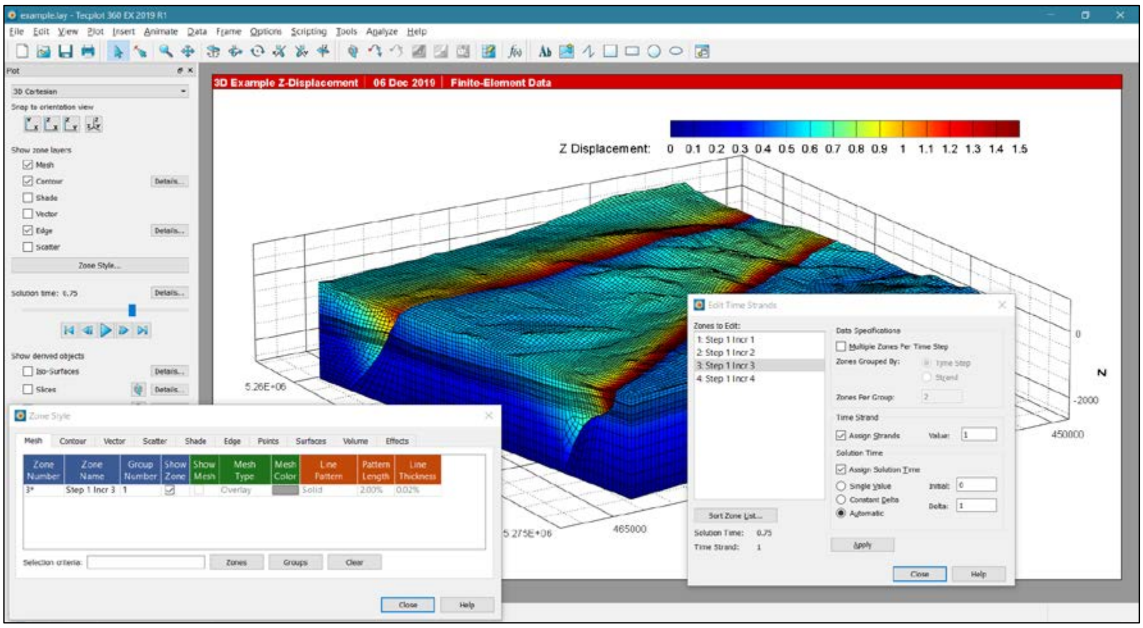
Show derived objects
☐ Iso-Surfaces
☐ Slices
☐ Streamtraces

Show effects
☐ Lighting
☐ Translucency

☑ Auto redraw

Fig. 2.2-1: Graphical user interface of Tecplot 360 EX.

The menu bar on the top offers the whole spectrum of functions within Tecplot 360 EX. The side bar on the left controls the plots generated.



Zone Number	Zone Name	Group Number	Show Zone	Show Mesh	Mesh Type	Mesh Color	Line Pattern	Pattern Length	Line Thickness
3*	Step 1 Incr 3	1	☑	☐	Overlay	▨	Solid	2.00%	0.02%

Zones to Edit:
1: Step 1 Incr 1
2: Step 1 Incr 2
3: Step 1 Incr 3
4: Step 1 Incr 4

Data Specifications
☐ Multiple Zones Per Time Step
Zones Grouped By: ● Time Step ○ Strand
Zones Per Group: 2

Time Strand
☑ Assign Strands Value: 1

Solution Time
☑ Assign Solution Time
○ Single Value Initial: 0
○ Constant Delta Delta: 1
● Automatic

Solution Time: 0.75
Time Strand: 1

Fig. 2.2-2: Relation between *Zone*, *Set of Zones*, *Time Strand* and *Solution Time*.

Each time increment of the Abaqus analysis is in Tecplot one *Zone*. Several *Zones* can be grouped into one a *Set of Zones* by assigning to them the same *Time Strand* value. In the example, all four increments have the value *Time Strand*=1 which means that they are displayed consecutively according to their *Solution Time*. The figure displays the increment 3 of the Abaqus step 1 with the *Solution Time* 0.75.

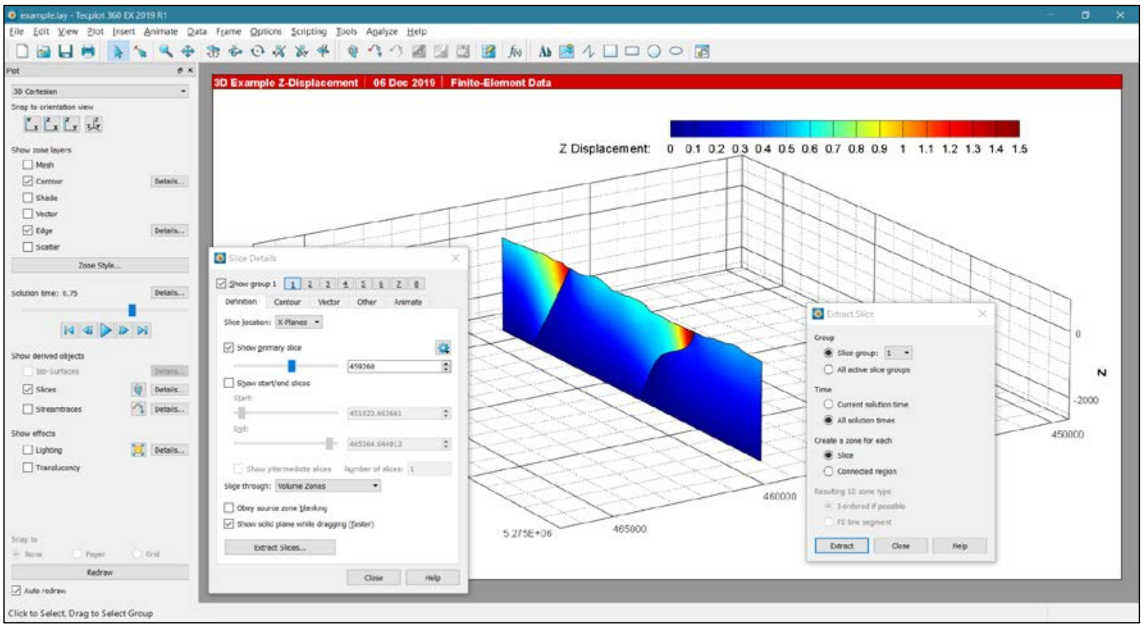


Fig. 2.2-3: Creation of a cross section through the model.

Use the *Slices* option from the side bar (turn off *Mesh*, *Contour* and *Edge* in the side bar) with options as indicated in the *Slice Details* dialog box on the left side.

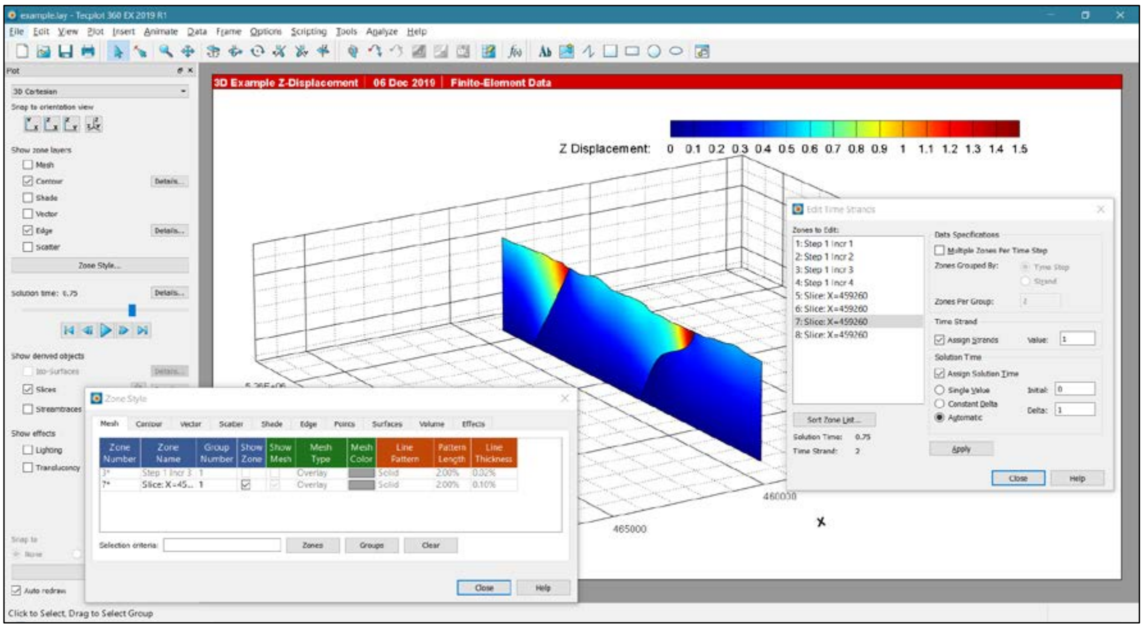


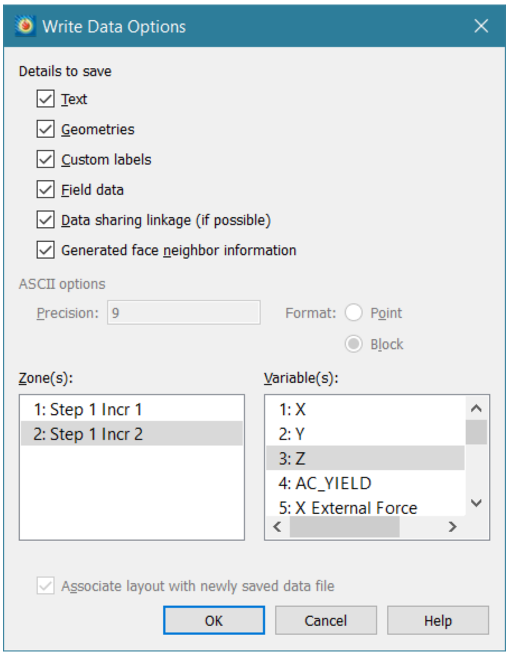
Fig. 2.2-4: Creation of a new *Set of Zones* and editing its *Time Strand* value.

Open the Zone Style… in the side bar and turn off the first *Set of Zones* (*Zones* 1-4). Then go to the menu bar and choose *Data-Edit Time Strands…* and you will see that you created four new *Zones* grouped into a new *Set of Zones* with a new (here 2) *Time Strand* value.

2.3 Tecplot 360 EX file formats **.lay* and **.plt*

After loading your model results (for details see section 3.3) the loaded data can be saved in the Tecplot 360 EX binary file format **.plt*. This is done with the menu bar *File/Write Data File…* and choosing there the data type section the *Tecplot Binary Data Writer (Current) (*.plt)* option. In the dialog box displayed in Fig. 2.3-1 the user can control which Abaqus steps, variables and other information from the binary **.odb*-file will be stored in the **.plt*-file format. In this **.plt-file* all model data and information e.g. on the finite element mesh (node and element numbers etc.) are stored.

The way the data are displayed on the screen is stored in the **.lay*-file which links to the **.plt*-file. Several views of the same dataset can be stored in several **.lay*-files. To save the current views settings of all frames, the menu bar offers to save the layout with *Save Layout* or *Save Layout As…*. The **.lay*-file contains the current view settings.



Write Data Options

Details to save

☑ Text
☑ Geometries
☑ Custom labels
☑ Field data
☑ Data sharing linkage (if possible)
☑ Generated face neighbor information

ASCII options

Precision: 9
Format: ○ Point
● Block

Zone(s):
1: Step 1 Incr 1
2: Step 1 Incr 2

Variable(s):
1: X
2: Y
3: Z
4: AC_YIELD
5: X External Force

☑ Associate layout with newly saved data file

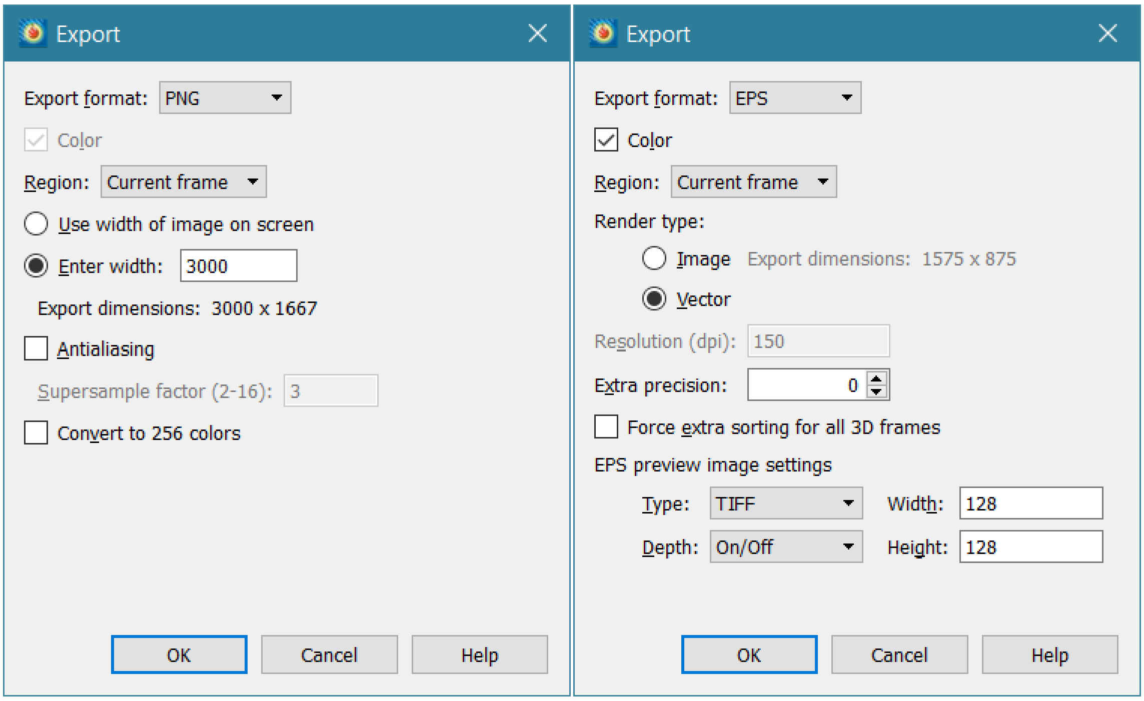
Fig. 2.3-1: Dialog box for saving the results in the Tecplot **.plt*-file format.

The user can select which *Zone* (Abaqus time step) is stored. Note that it is recommended to link the current *.lay-file with the **.plt*-file with the tick box on the bottom.

2.4 Tecplot 360 EX output options

2.4.1 Figure formats

Saving the active frame as an image the option *File/Export…* from the menu bar is used. Here the user can choose between four raster image formats (png, tiff, jpeg, eps) and two vector image formats (wmf, eps). For the *.eps-format one can either store the file as an image with the wanted resolution in dots per inch (dpi) or as a vector format (Fig. 2.4-1). The two vector formats (wmf and eps-vector) can be quite large e.g. for 3D views of the model results including the mesh. Each line and each element face displayed on the 3D view then is one vector.



Export format: PNG
☑ Color
Region: Current frame
○ Use width of image on screen
● Enter width: 3000
Export dimensions: 3000 x 1667
☐ Antialiasing
Supersample factor (2-16): 3
☐ Convert to 256 colors

Export format: EPS
☑ Color
Region: Current frame
Render type:
○ Image Export dimensions: 1575 x 875
● Vector
Resolution (dpi): 150
Extra precision: 0
☐ Force extra sorting for all 3D frames
EPS preview image settings
Type: TIFF
Width: 128
Depth: On/Off
Height: 128

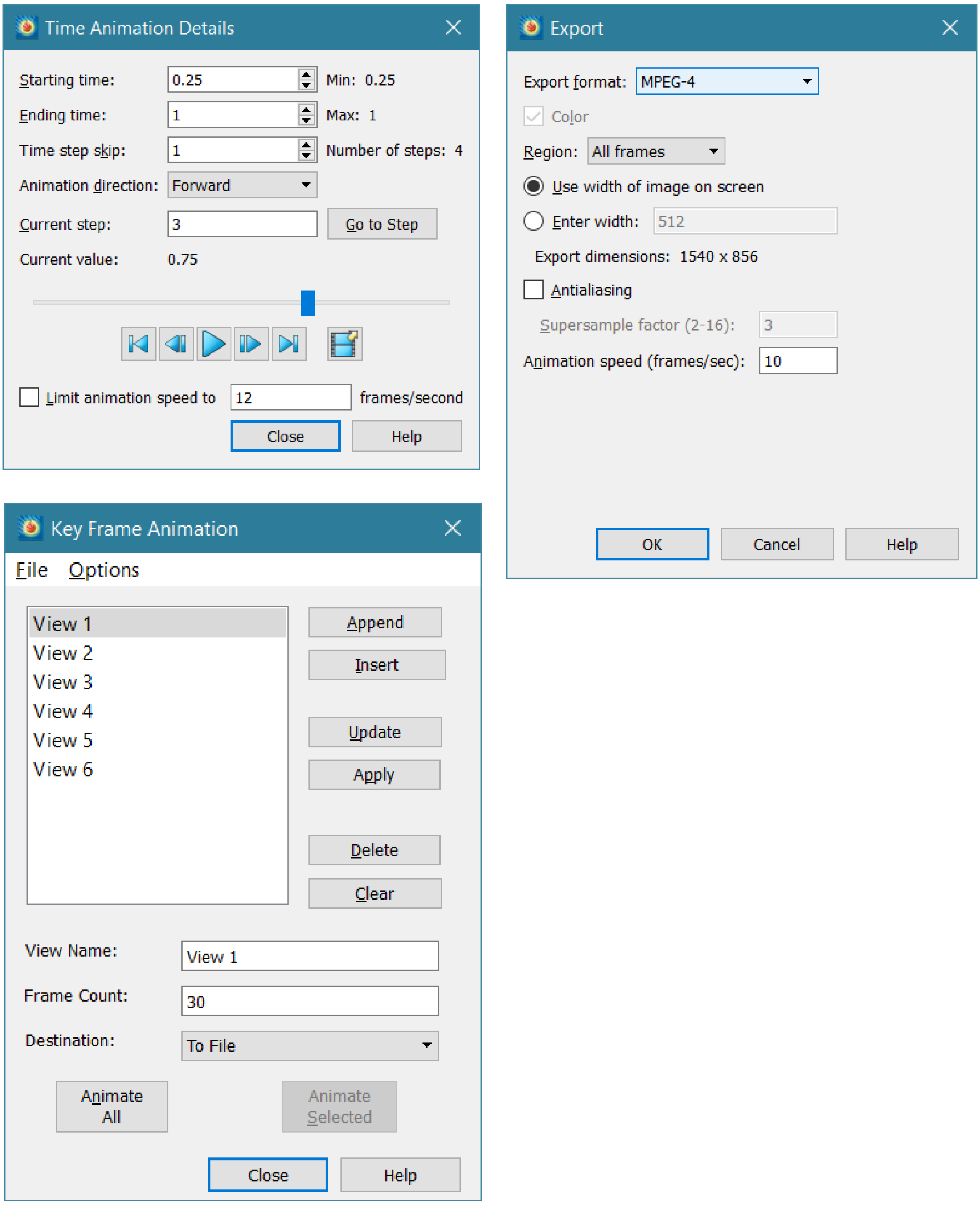
Fig. 2.4-1: Dialog boxes for saving model results as a figure format **.png* or **.eps*.
Note that the default setting of the figure resolution is the width of the screen. To enhance the quality enter the width (here 3000 pixel). Note that you can choose for the postscript output **.eps* both, a raster image or a vector format. The vector format can be quite extensive e.g. when you save a 3D view including the mesh information.

2.4.2 Movie formats

The model results can also be saved as movies in different formats (Fig. 2.4-2). There are two different ways to generate a movie. Either the active frame is animated over the *Solution Time* (*Animate/Time…* in the menu bar) or different views (e.g. rotate the 3D plot view of the model result) within the active frame are saved and Tecplot 360 EX interpolates the movement between these views (*Animate/Key Frame Animation…* in the menu bar).

The two consecutive appearing dialog boxes for the first approach are displayed in Fig. 2.4-2 on the left and the centre dialog boxes. When you press the “film” button the dialog box on right top of Fig. 2.4-2 appears where the movie output format and the individual setting of the format can be controlled. Note that each time step is a *Zone* within Tecplot 360 EX. The model time is stored in the variable *Solution Time* within Tecplot 360 EX. Its content is given by the time defined in the Abaqus model which is in general given in seconds. In elastic calculation, it is not a time, but an arbitrarily number given to each increment.

The other way to generate a movie is to store consecutively several different views of the activate frame within the dialog box on right of Fig. 2.4-2. Tecplot 360 EX interpolates the movement between the different frames and generates a movie that is either saved in a file or displayed on the screen.



Time Animation Details

Starting time: 0.25 Min: 0.25

Ending time: 1 Max: 1

Time step skip: 1 Number of steps: 4

Animation direction: Forward

Current step: 3

Current value: 0.75

☐ Limit animation speed to 12 frames/second

Export

Export format: MPEG-4

☑ Color

Region: All frames

● Use width of image on screen

○ Enter width: 512

Export dimensions: 1540 x 856

☐ Antialiasing

Supersample factor (2-16): 3

Animation speed (frames/sec): 10

Key Frame Animation

View Name: View 1

Frame Count: 30

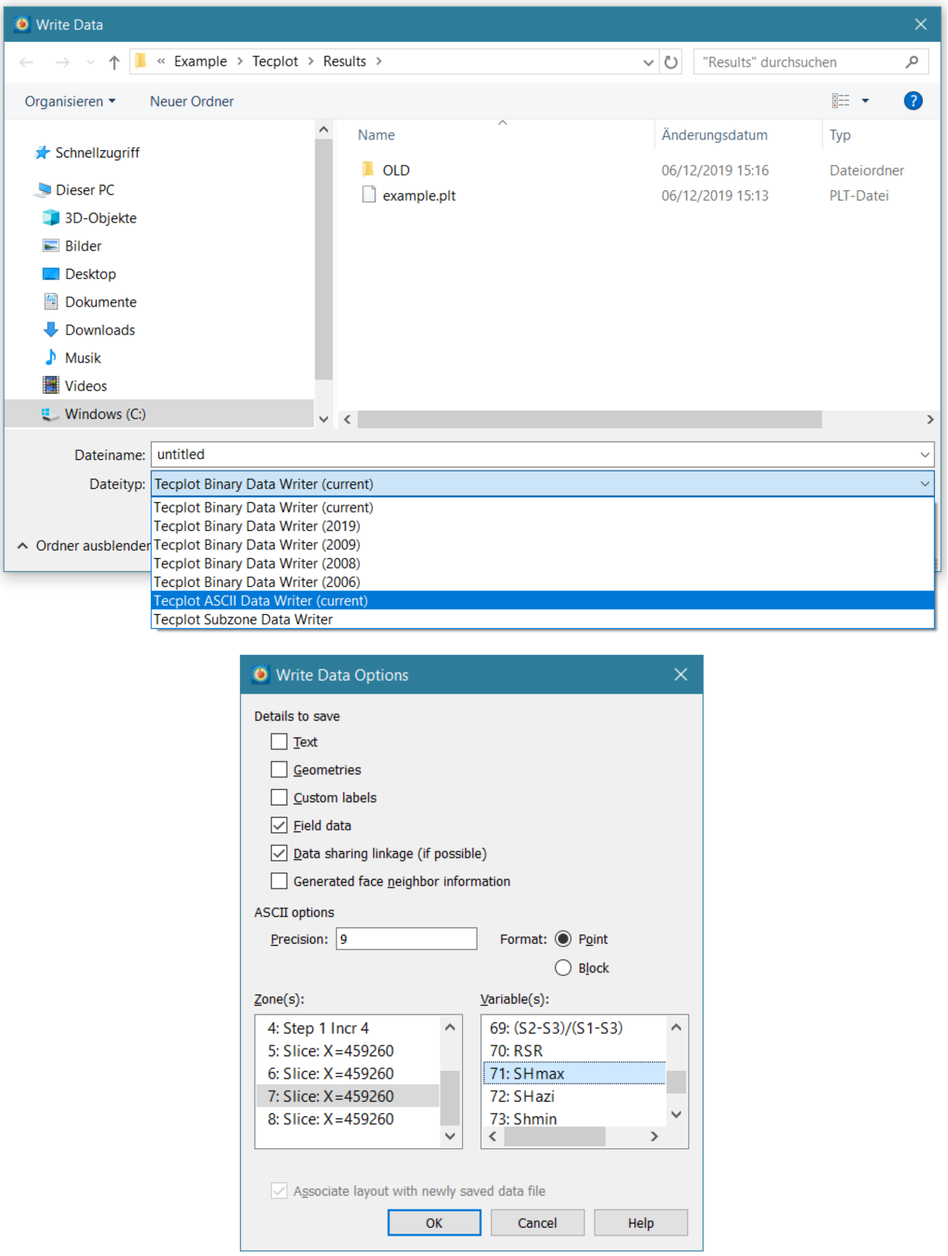
Destination: To File

Fig. 2.4-2: Dialog boxes for saving model results as a movie.

Left/Centre: When *Destination To File* is selected the dialog box in the centre appears where format and other movie settings are determined. Right: The alternative way to generate a movie is the interpolation between different views in the current frame.

2.4.3 ASCII data output format

Independent from the current active frame or *Zone* the model results can be exported into an ASCII file format using the menu bar *File/Write Data File…*. Choose the data type *Tecplot ASCII Data Writer (current) (*.dat)* and provide an output data file name. With the resulting dialog box (Fig. 2.4-3) the user can choose from which *Zone* (e.g. a certain slice at a certain time step of the model; see Fig. 2.4-3) which variable is written to the ASCII file. To create a line-wise output of the data choose the *Format Point* toggle.



Name	Änderungsdatum	Typ
OLD	06/12/2019 15:16	Dateiordner
example.plt	06/12/2019 15:13	PLT-Datei

Dateiname: untitled
Dateityp: Tecplot Binary Data Writer (current)

Details to save
☐ Text
☐ Geometries
☐ Custom labels
☑ Field data
☑ Data sharing linkage (if possible)
☐ Generated face neighbor information

ASCII options
Precision: 9
Format: ● Point
○ Block

Zone(s):
4: Step 1 Incr 4
5: Slice: X=459260
6: Slice: X=459260
7: Slice: X=459260
8: Slice: X=459260

Variable(s):
69: (S2-S3)/(S1-S3)
70: RSR
71: SHmax
72: SHazi
73: Shmin

☑ Associate layout with newly saved data file

Fig. 2.4-3: Dialog boxes to save selected model results in ASCII data format.
Note that the format in the dialog box Point has to be selected in order to receive a line-wise format that can be easily used in other visualization tools.

2.5 Using frame styles and macros

There are three options to generate and load very fast a certain view on your data for a pre-defined variable for specific increment of a specific *Zone*. As explained in section 2.3, the layout file **.lay* stores the actual view on your data as it appears on the screen using the menu bar *File/Save Layout As...* The view can contain several frames from different increments and set of *Zone*s and it links the view parameter to the **.plt* file where the model results are stored in the Tecplot 360 EX 2019 R1 format. In the following two further options are presented that accelerate the generation of a specific view and/or data analysis using frame styles or macros.

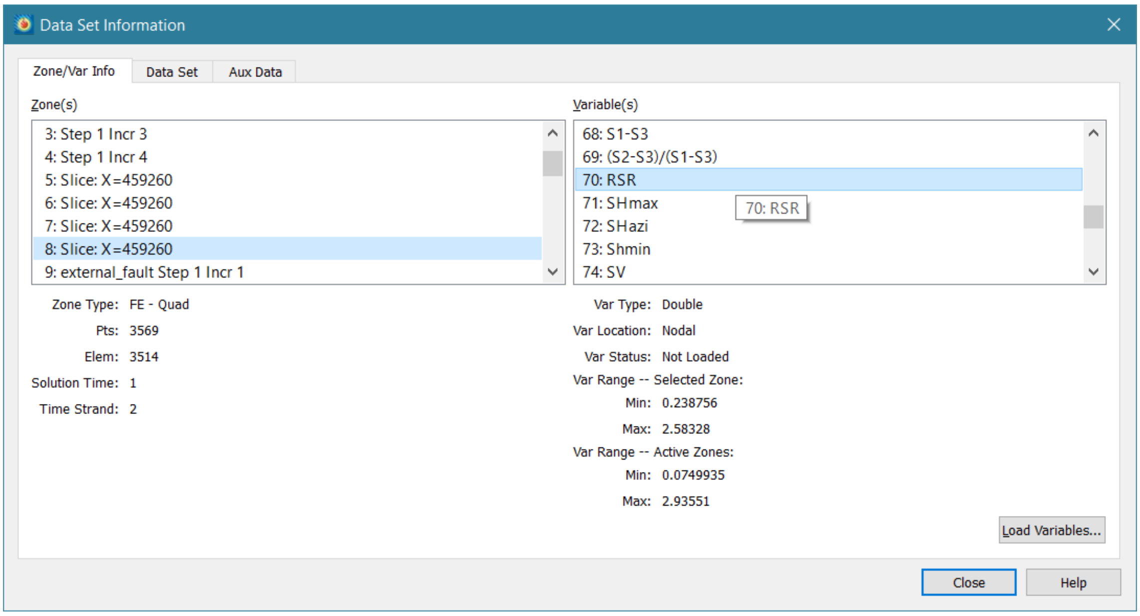
2.5.1 Frame Styles

A view of a single frame (the active one) can be stored in the so-called *Frame Style* file **.sty.* When such a file is loaded and applied to another model the stored view and analysis settings are applied. It serves as a template to reproduce the view with all details of the stored setting. An example is given in the file with the name *3D_View_Z_Displacement.sty*. However, the displayed variable – here the z component of the displacement - is not called with the variable name in the frame style description, but with the variable number that is attached to each model variable.

The following lines show the first lines of the **.sty* file. The command to generate a contour plot $!GLOBALCONTOUR 1 is followed in the next line by the option *VAR=49* which is variable number but not the name of the variable.

```
#!MC 1410
$!SETSTYLEBASE FACTORY
$!PLOTTYPE = CARTESIAN3D
$!FRAMELAYOUT
.
.
.
$!GLOBALCONTOUR 1
 VAR = 49
 COLORMAPNAME = 'Modified Rainbow - Dark ends'
 DEFNUMLEVELS = 16
```

The numbering of the variables can differ and depends on the Tecplot 360 EX analysis procedure of the user. Thus, when another model is loaded, but the order of the data analysis is different, the numbering of the variables changes as well and the **.sty*-file would not produce the correct view results as it uses strictly the variable number that might have changed. The only way to avoid this is to load the individual model runs always in the same way and to generate in the same order the variable with *GeoStress*. In the dialog box from the menu bar *Data/Data Set Info….* Tecplot 360 EX provides the user with the variable numbers that are assigned to each variable name (Fig. 2.5-1). When the number of the created variable is not correct anymore, the user can edit the *Frame Style* by changing the number.



Zone(s)

3: Step 1 Incr 3
4: Step 1 Incr 4
5: Slice: X=459260
6: Slice: X=459260
7: Slice: X=459260
8: Slice: X=459260
9: external_fault Step 1 Incr 1

Zone Type: FE - Quad
Pts: 3569
Elem: 3514
Solution Time: 1
Time Strand: 2

Variable(s)

68: S1-S3
69: (S2-S3)/(S1-S3)
70: RSR
71: SHmax
72: SHazi
73: Shmin
74: SV

70: RSR

Var Type: Double
Var Location: Nodal
Var Status: Not Loaded
Var Range -- Selected Zone:
Min: 0.238756
Max: 2.58328
Var Range -- Active Zones:
Min: 0.0749935
Max: 2.93551

Fig. 2.5-1: Data set information dialog box.

Note that the numbers assigned to the model variable can change from one model to the next. The numbers are given in the order the variables are loaded or generated within Tecplot 360 EX (and the Add-on *GeoStress*). Below the list the range of the values of the selected variable in the selected zone is shown. Here it is the *RSR* value in the fourth increment on the *Slice: X=459260*.

2.5.2 Macros

A macro is quite similar to a *Frame Style*, but even more powerful. The user can insert any commands that are provided by Tecplot 360 EX to analyse and display the model results. Using the menu bar *Scripting/Record Macro....* Tecplot 360 EX records all steps you are performing until you finish the recording. The generated **.mrc* file looks very similar to the **.sty* files as it uses the same header and same Tecplot 360 EX commands. In the example provided with the name *macro_example_RSR.mrc* the view from the z displacement view in the previous example is changed to a 3D view of the RSR value. Again Tecplot 360 EX is using the variable number instead of the variable name (RSR has the variable number 71 in the example).

```
#!MC 1410
$!VarSet |MFBD|='C:\Program Files\Tecplot\Tecplot 360 EX 2019 R1'
$!PICK ADDATPOSITION
 X = 3.77792915531
 Y = 1.37261580381
 CONSIDERSTYLE = YES
$!SETCONTOURVAR
 VAR = 71

.
.
$!RemoveVar |MFBD|
```

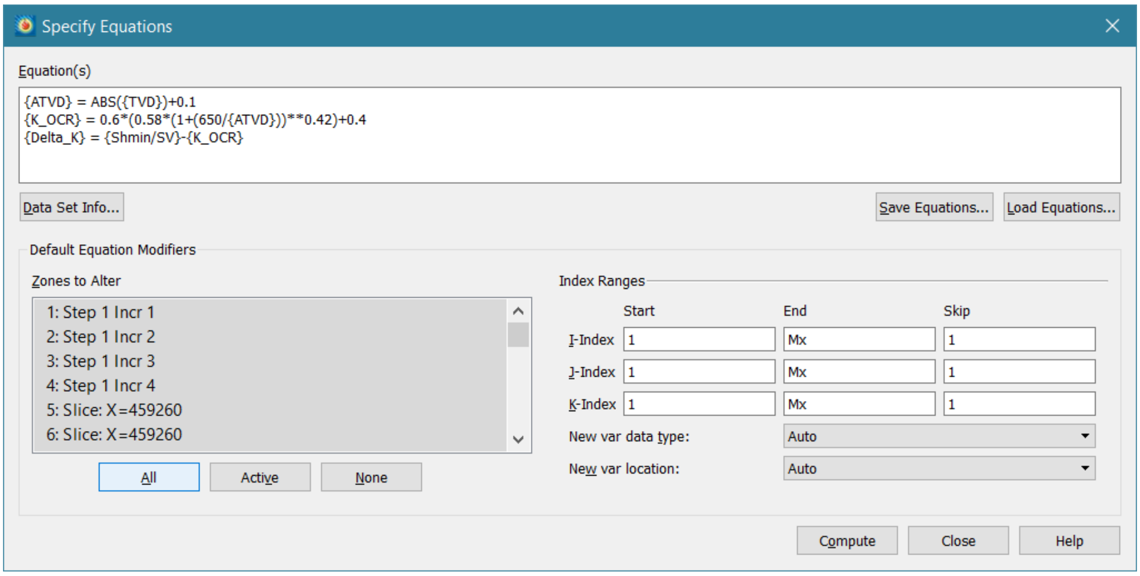
However, in contrast to the *.sty file the choice of the variable can be changed using instead of the line VAR = 71 the command *$!GETVARNUMBYNAME |MyVar| NAME="RSR"*. This edited version of the macro is provided in the file *macro_example_RSR_edited.mcr*. Note the macro is using the variable name instead of the variable number. Hence it is not dependent on the exact recreation of the data loading.

2.6 Calculation of new variables with the equation tool

From the existing variables in the model the user can calculate further variables using the menu bar *Data/Alter/Specify Equations…*. The user can either type in the equations in the dialog box that appears (Fig. 2.6-1) or load the equations from a **.eqn*-file. Examples are provided in the two files *delta_K.eqn* and *displacements.eqn*.

An example of the format is given below. Variables that exist in the model are stated in curly brackets. Tecplot 360 EX also provides a wide range of functions that can be integrated in the equations. Note that it is possible to alter variables in only one *Zone* (append the *Zone* number in square brackets) but it is not possible to create a new variable in only one *Zone*. More details are given in the Tecplot 360 EX user manual.

```
#!MC 1400
# Created by Tecplot 360 build 14.0.2.33360
$!ALTERDATA
 EQUATION = '{ATVD} = ABS({TVD})+0.1'
$!ALTERDATA
 EQUATION = '{K_OCR} = 0.6*(0.58*(1+(650/{ATVD}))**0.42)+0.4'
$!ALTERDATA
 EQUATION = '{Delta_K} = {Shmin/SV}-{K_OCR}'
```



```
{ATVD} = ABS({TVD})+0.1
{K_OCR} = 0.6*(0.58*(1+(650/{ATVD}))**0.42)+0.4
{Delta_K} = {Shmin/SV}-{K_OCR}
```

Zones to Alter:
1: Step 1 Incr 1
2: Step 1 Incr 2
3: Step 1 Incr 3
4: Step 1 Incr 4
5: Slice: X=459260
6: Slice: X=459260

	Start	End	Skip
I-Index	1	Mx	1
J-Index	1	Mx	1
K-Index	1	Mx	1

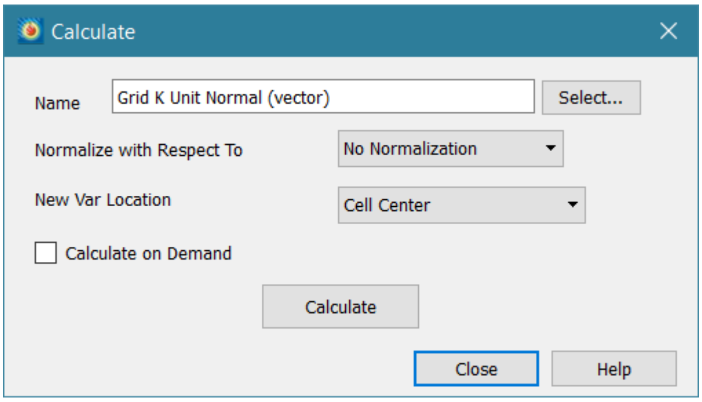
New var data type: Auto
New var location: Auto

Fig. 2.6-1: Write Data as Text File dialog box.

Note that the user can control for which *Zone* (increment, geometry) which variable is written to the ASCII output file.

2.7 Calculation of the normal vector of surfaces

The following workflow presents how to generate and visualize the normal vector of surfaces in the model. Go to the menu Analyse/Calculate Variables and select there the function Grid K Unit Normal (Vector) in the appearing dialog box (Fig. 2.7-1). Choose for the *New Var Location = Cell Center* and turn off the button *Calculate on Demand* in order to store the calculated variable. Press the button Calculate in order to calculate the three components *X Grid K Unit Normal*, *Y Grid K Unit Normal*, *Z Grid K Unit Normal* of the unit vector of all geometries (Fig. 2.7-2).



Name: Grid K Unit Normal (vector)

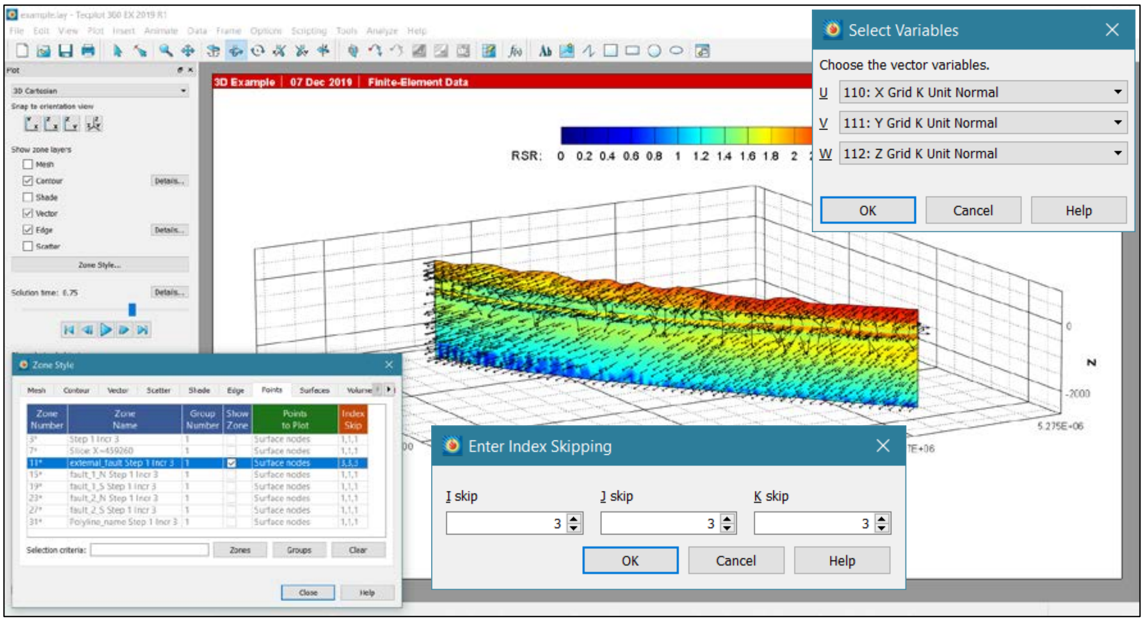
Normalize with Respect To: No Normalization

New Var Location: Cell Center

☐ Calculate on Demand

Fig. 2.7-1: Dialog box for calculation of variables.

This tool box offers a huge range of functions that can be applied. For further details see the Tecplot 360 EX manual.



Select Variables

Choose the vector variables.

U: 110: X Grid K Unit Normal

V: 111: Y Grid K Unit Normal

W: 112: Z Grid K Unit Normal

Enter Index Skipping

I skip: 3

J skip: 3

K skip: 3

Fig. 2.7-2: Visualization of the normal vector of surfaces.

Upper right dialog box shows how to choose the three components of the vector to be displayed. As the number of vectors can be rather large Tecplot 360 EX offers the possibility to display a reduced number of vectors. This is controlled in the *Zone Style…*. on the page Points with the Index Skip. The dialog box on the right bottom shows the I and K direction in which only every third vector should be displayed.

2.8 Using PyTecplot

With Tecplot 360 2017 R3 PyTecplot was released which is a powerful automatization tool. In addition to the control of Tecplot with a macro language, PyTecplot allows the direct access to files loaded in Tecplot from within a Python code. In comparison to macros the full functionality of PyTecplot is not yet achieved. However, PyTecplot supports the execution of macro commands and thus the functionality from existing macros can be integrated and combined with the computation power of a real programming language.

Below, we show only a very simple example that exemplifies the power of PyTecplot. It shows the following four steps:

1. Loading of a dataset. The direct loading of Abaqus output files is not (yet) possible but workarounds exist.
2. Execution of an equation converts the XX Stress component from compression-negative Pascal to compression-positive Mega-Pascal.
3. Probing, extraction, and display of a Tecplot variable at a certain location with Python in four steps:
    - The index of the Tecplot variable XX Stress geol is looked up.
    - All variables at a certain location in zone 1 (due to Pythons zero-based indexing here zone 0) are extracted to a Python variable.
    - From the Python variable *result* the value for the XX Stress is extracted.
    - The newly generated Python variable XX_Stress_geol is printed to the screen.
4. Saving of the update dataset into a new **.plt*-file

```
# The Tecplot library is imported.
import tecplot

# The dataset example.plt is loaded.
dataset = tecplot.data.load_tecplot('hipy_test_result.plt')

# Execution of an equation.
tecplot.data.operate.execute_equation('{XX Stress geol} = {XX Stress}*-1e-6')

# Probing and extraction of a Tecplot variable for Python.
tecvar = dataset.variable('XX Stress geol').index

result = tecplot.data.query.probe_at_position(5000,6000,-2500,zones=[0])

XX_Stress_geol = result[0][tecvar]*(-1e-6)

print XX_Stress

# The Tecplot dataset is saved to a new .plt file.
tecplot.data.save_tecplot_plt("%s"%('example_new.plt'))
```

This example does not indicate the numerous benefits of PyTecplot when it comes to the control and automatization of layout files or the usage of Python loops. A full documentation of the PyTecplot library is included in the Tecplot installation and should be referred to for any detailed inquiries on the functionality.

3 The Add-on *GeoStress* v2.0

3.1 Installation of the Add-on *GeoStress*

The Add-on *GeoStress* is a shared library compiled and tested on 64-bit Windows platforms (*GeoStress.dll*) for the release Tecplot 360 EX 2019 R1. To install the add-on the appropriate library file has to be copied into the execution directory of Tecplot 360 (*../bin*). Furthermore, you have to edit the file *tecplot.add* that is located in the Tecplot 360 EX installation directory. The following lines must be added at the end of this file:

```
$!LoadAddon "GeoStress”
```

In Tecplot 360 EX you will find the Add-on *GeoStress* under the menu bar *Extras*.

The Add-on for 64-bit Linux platforms is provided with the shared library *libGeoStress.so. Just like under Windows, this file must be placed in the Tecplot execution directory (../bin). The corresponding entry in the Add-on configuration file tecplot.add is*

```
$!LoadAddon "libGeoStress”
```

3.2 Basics of the Add-on *GeoStress*

The Add-on *GeoStress* loads the main geometry data of the model (total number of nodes and elements) from the first *Zone*. Even though Tecplot 360 EX allows you to append a second model data set with different number and/or numbering of nodes and elements the Add-on *GeoStress* would not recognize this second data set.

GeoStress needs the following variables from the output file of the finite element solver: The three geometry variables for the location of the element nodes (X, Y, Z), the six components of the stress tensor (*XX Stress*, *YY Stress*, *SZZ Stress*, *SXY Stress*, *SYZ Stress*, *SZX Stress*) and the three components of the displacement vector at the nodes (*X Displacement*, *Y Displacement*, *Z Displacement*). Note that the components of the stress tensor in commercial finite element packages are not provided at the nodes, but at the integration points of the elements. Tecplot 360 EX has to interpolate these using the element description of the individual element types of the software packages. The current version Tecplot 360 EX 2019 R1 can handle the Abaqus version 2019 binary odb-output file. However, note that newer Abaqus versions may be not supported by Tecplot. For other commercial finite element software packages we did not test which version can be loaded, but this information is given in the Tecplot 360 EX manuals.

> **Note** that *GeoStress* cannot be used for 2D model results since most calculations expect the six components of the 3D stress tensor.

The execution of the Add-on *GeoStress* either adds further variables to the existing first *Set of Zones* or it creates a new *Set of Zones* and interpolates all variables to this new *Set of Zones*. The recommended general workflow using *GeoStress* is the following:

1. Import the Abaqus odb-file (or the output file from your software or use the general data loader in the case your software output file is not supported).
2. Generate with *GeoStress* all required variables in the first *Set of Zones*.
3. Generate new *Set of Zones* (e.g. cross sections or external surfaces) either with *GeoStress* or using the functionality of Tecplot 360 EX.

Note that the generation of new Set of Zones should occur after all required variables are generated with *GeoStress*. Otherwise, Tecplot may run into problems due to variables that are present in some, but not all zones.

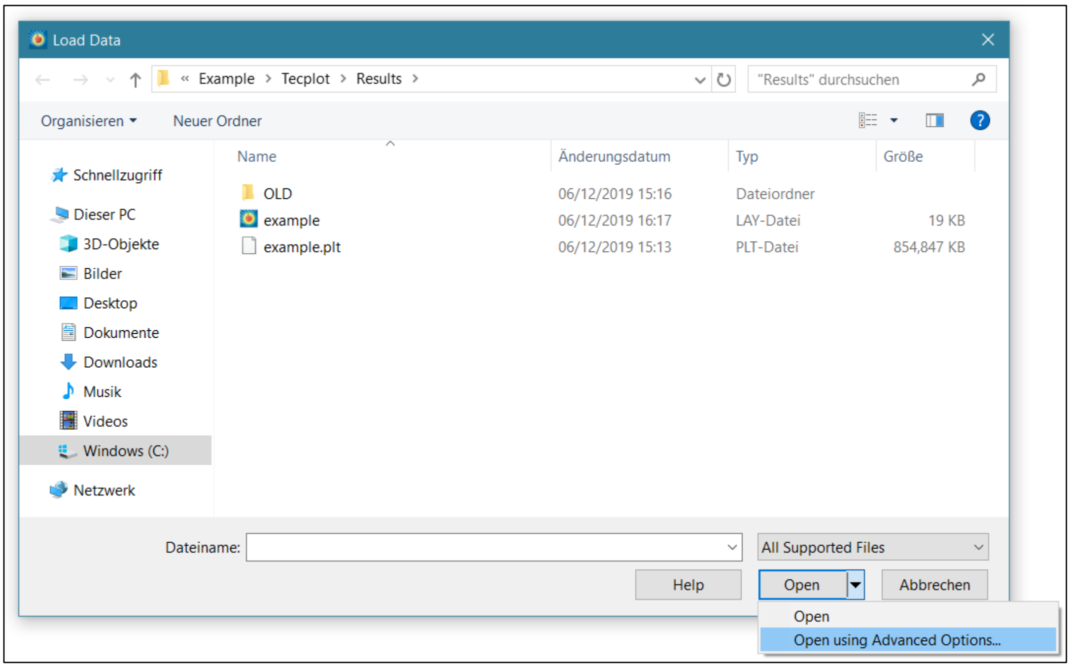
3.3 Loading of Abaqus output files

Tecplot is not limited to Abaqus output files, but as we provide an example using the binary output file of Abaqus, we explain in the following the loading options for this file format in more detail. In principal, the Add-on *GeoStress* should work with all kind of finite element output files, but we did not test this. As stated in section 3.2, the basic information that is expected from *GeoStress* are the three coordinates of the nodes (*X*, *Y*, *Z*), the six components of the stress tensor (*SXX*, *SYY*, *SZZ*, *SXY*, *SXZ*, *SYZ*) and the three components of the displacement vector (*X displacement*, *Y displacement*, *Z displacement*). Tecplot 360 EX can also load the output files from other commercial finite element software packages such as Nastran or Ansys. For other finite element solver a different import filter can be used.

Using Abaqus, Tecplot can load three different file types: The **.fil* and the **.odb* output file format as well as the **.inp* input file format from Abaqus, e.g. in order to import into the analysis further geometries. Running on a Linux system, Tecplot is only able to read **.fil* files which are not created by Abaqus per default. On the lower right part the dialog box (Fig. 3.3-1) for loading odb-files offers to use advanced options. When this option is used an additional dialog box (Fig. 3.3-2) comes up where the option *Subdivide Zone by Components* or *by Elements Type* can be set. This option will put components and/or all elements with the same material property assignments for each time step into a separate *Zone*. This would be comfortable since e.g. individual lithological layers can be displayed individually over time, but *GeoStress* expects that all elements and nodes that exist in the model are defined in the first *Set of Zones* of the model data set (see also section 3.2 for further details) and thus this selection can lead to inconsistencies of even major problems while using *GeoStress*. Thus, this option has to be used carefully and is not recommended.

The second option in the dialog box (Fig. 3.3-1) is on *Select Zones and Variables*. With this option specific variables or increments/steps can be deselected (default setting is all variables and all increments and steps (Fig. 3.3-2).

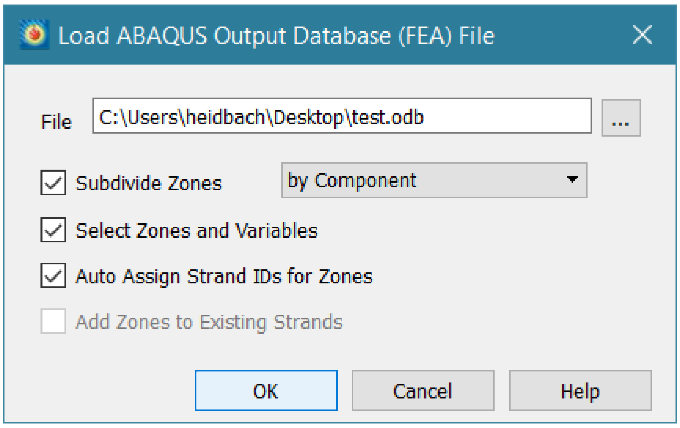
After applying the options set in Fig. 3.3-1 and the subsequent dialog box displayed in Fig. 3.3-3 another dialog box appears (Fig. 3.3-4). Here the user is asked to change settings on the plot scaling factor.



Name	Änderungsdatum	Typ	Größe
OLD	06/12/2019 15:16	Dateiordner	
example	06/12/2019 16:17	LAY-Datei	19 KB
example.plt	06/12/2019 15:13	PLT-Datei	854,847 KB

Fig. 3.3-1: Loading Abaqus odb-files.

Default loading is without advanced loading options. It is recommended not use the advanced loading options to avoid problems that may arise during the analysis using *GeoStress* (see text for further explanation).



File: C:\Users\heidbach\Desktop\test.odb

☑ Subdivide Zones — by Component

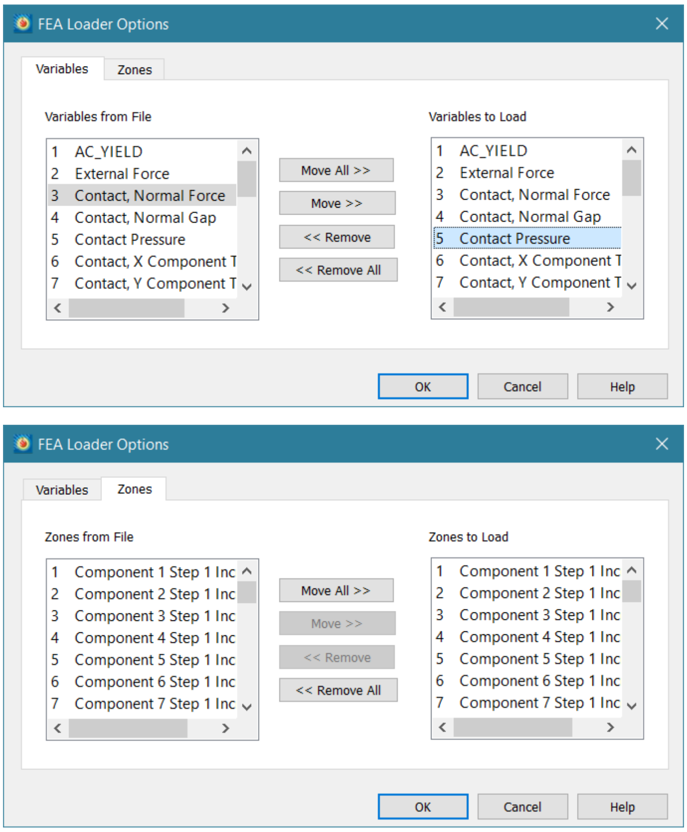
☑ Select Zones and Variables

☑ Auto Assign Strand IDs for Zones

☐ Add Zones to Existing Strands

Fig. 3.3-2: Advanced loading options for Abaqus odb-files – part I.

When *Subdivide Zone* is selected the elements with the same material property assignments (each element set) are put for each time step into a separate *Set of Zones* leading to limits with the Add-on *GeoStress* as it expects that the whole model is located in the first *Set of Zones*.



Variables from File
1 AC_YIELD
2 External Force
3 Contact, Normal Force
4 Contact, Normal Gap
5 Contact Pressure
6 Contact, X Component T
7 Contact, Y Component T

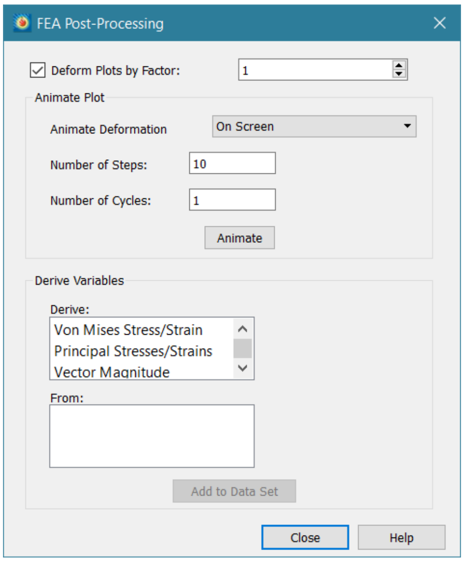
Variables to Load
1 AC_YIELD
2 External Force
3 Contact, Normal Force
4 Contact, Normal Gap
5 Contact Pressure
6 Contact, X Component T
7 Contact, Y Component T

Zones from File
1 Component 1 Step 1 Inc
2 Component 2 Step 1 Inc
3 Component 3 Step 1 Inc
4 Component 4 Step 1 Inc
5 Component 5 Step 1 Inc
6 Component 6 Step 1 Inc
7 Component 7 Step 1 Inc

Zones to Load
1 Component 1 Step 1 Inc
2 Component 2 Step 1 Inc
3 Component 3 Step 1 Inc
4 Component 4 Step 1 Inc
5 Component 5 Step 1 Inc
6 Component 6 Step 1 Inc
7 Component 7 Step 1 Inc

Fig. 3.3-3: Advanced loading options for Abaqus odb-files – part II.

When the option *Select Variables and Zones* is selected, the user can determine which variables and steps/increments are loaded.



☑ Deform Plots by Factor: 1

Animate Deformation: On Screen

Number of Steps: 10

Number of Cycles: 1

Derive:
Von Mises Stress/Strain
Principal Stresses/Strains
Vector Magnitude

From:

Fig. 3.3-4: Advanced loading options for Abaqus odb-files – part III.

Here the plot deformation can be set to exaggerate small deformation Even though this is interesting in geosciences it is not recommended to change this. See subsection 3.1.4 for details and explanations.

3.4 Using *GeoStress* with PyTecplot

To allow a full automatization of the analysis and visualization the libraries *GeoStressCmd.dll* (for Windows) and *libGeoStressCmd.so* (for Linux) have been developed for usage with macros or PyTecplot. The libraries contain the same functionality as the Add-on *GeoStress,* but they have no graphical user interface. They can only be accessed via macro commands or PyTecplot. In both cases *GeoStressCmd* is executed by an “extended command” and controlled by a command string. The command string that controls *GeoStressCmd* is identical for the usage with a macro or PyTecplot.

First, the according library has to be initialized by a string variable that is equivalent to the name of a *GeoStress* tab. Second, some general settings have to be initialised (see 4.1). Third, the specific functions (see 4.2 – 4.7) of *GeoStressCmd* and variables that are to be executed are specified according to the radio buttons in the GUI by a ‘1’ (compute) or a ‘0’ (do not compute). Additionally, individual information has to be provided for some of the functionalities, such as zone names, fault orientation and properties, or external geometry files. Abbreviations for the syntax, explanations, and examples are summed up in Table 1.

The individual command strings can be found in the following description of *GeoStress/GeoStressCmd* functionalities. For each function, a reference command string with the abbreviations from Table 3.4-1 and an example are provided.

GeoStressCmd: It is executed individually with an individual command string for each “tab” of *GeoStress*. In a macro file *GeoStressCmd* execution is realised with an extended command, exemplified by:

```
$!EXTENDEDCOMMAND
  CommandProcessorID = "GeoStressCmd"
  Command = "Stress, 90.0, 0.0, 0.0, 0.0, -1.0e-6, 1, 1, 1, 1, 0, 0, 0, 0, 1"
```

In PyTecplot an extended macro command is executed with the same command string but a slightly different Python code. This is exemplified by:

```
CommandString = "Stress, 90.0, 0.0, 0.0, 0.0, -1.0e-6, 1, 1, 1, 1, 0, 0, 0, 0, 1"
tecplot.macro.execute_extended_command("GeoStressCmd",CommandString)
```

Tab. 3.4-1: Information in the command string.

Not all information is required for each function.

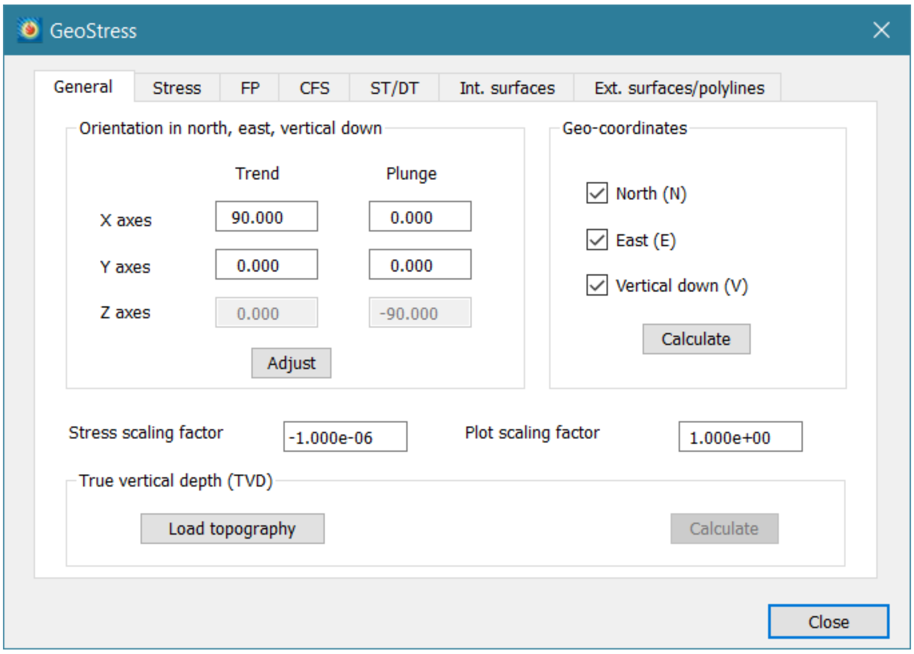
Abbreviation	Information	Example (Range)
xtr	Trend of the x-axis in the model in degree	90 (0° - 360°)
xpl	Plunge of the x-axis in the model in degree	0 (0° - 90°)
ytr	Trend of the y-axis in the model in degree	0 (0° - 360°)
ypl	Plunge of the y-axis in the model in degree	0 (0° - 90°)
ssf	Stress scaling factor	-1e-6
psf	Plot scaling factor	1
fr	Friction angle in degree	30
co	Cohesion in MPa	10
te	Tensile strength of the rock in MPa	5
fst	Fault strike	55 (0° - 360°)
fdp	Fault dip	50 (0° - 90°)
frk	Fault rake	-80 (-180° - 180°)
zn1, zn2	Zone names (case sensitive)	Step 1 Incr 4
sn1, sn2	Surface names (case sensitive)	fault_1_N
fn	Full path to file and filename	C:\\Desktop\\Tecplot_Example\\external_polyline.txt
nx	Radio button equivalent	1 (0, 1)

4 User interface of the Add-on *GeoStress*

The user interface of the Add-on *GeoStress* is opened in the menu bar under *Extras* and appears as a dialog box with seven different tab cards (Fig. 4.1-1). The dialog box of the Add-on *GeoStress* starts with the page *General* which is in particular important during the first application of *GeoStress* on a newly loaded model output file. The other six tab cards of *GeoStress* group the functions regarding values derived from the stress tensor, Fracture Potential (*FP*), Coulomb Failure Stress (*CFS*), Slip and Dilation Tendency (*ST* and *DT*), Internal Surfaces and External Surfaces as well as Polylines. Details and handling of these tab cards are described in the following sections 4.1-4.7 of this chapter. In the first five tab cards, new variables are calculated and provided for the first *Set of Zones*. Only the options in page six and seven generate both new variables and new *Zone* (*Set of Zones*).

4.1 General settings

The first tab of the Add-on *GeoStress* is the General Setting. This tab has five sections (Fig. 4.1-1). In particular, the setting of the stress and plot scaling factors is important as they have an impact on the variable calculation in the other tabs.



Orientation in north, east, vertical down

	Trend	Plunge
X axes	90.000	0.000
Y axes	0.000	0.000
Z axes	0.000	-90.000

Geo-coordinates

☑ North (N)
☑ East (E)
☑ Vertical down (V)

Stress scaling factor -1.000e-06
Plot scaling factor 1.000e+00

Fig. 4.1-1: Page General.

The five sections of this page control the orientation of the coordinate system, the stress and plot scaling factor and the calculation of the variable True Vertical Depth (TVD).

GeoStressCmd: In GeoStressCmd the according general settings have to be included in each individual command string. Depending on the function not all information provided in the general settings are required in GeoStressCmd. In the following sub-sections the GeoStressCmd command strings are stated at the end of each sub-section.

4.1.1 Orientation in North, East and Vertical down

Most of the new variables that are calculated with *GeoStress* require a reference to a geo-orientated coordinate system {*N* (north), *E* (east), *V* (vertical down)}. The transformation from the modelling system {*X*, *Y*, *Z*} to {*N*, *E*, *V*} is described by the trend and plunge of the *X*-axis, *Y*-axis and *Z*-axis respectively in the geo-system. The trend is given as a 360° angle measured clockwise from *N*. The plunge can be between -90° and 90°. A positive plunge indicates that an axis points downwards (Fig. 4.1-2).

The default orientation is *X*=*E* (90; 0), i.e. the trend of the *X*-axis is 90° from the North direction, *Y*=*N* (0; 0), i.e. the trend of the *Y*-axis is the North direction and *Z*=-*V* (0; -90), i.e. z-values downwards are positive. The Add-on offers to re-orient the coordinate system, i.e. to re-orient the six components of the 3D stress tensor and to make sure that the positive *X*-, *Y*-, *Z*-axes are oriented in North (*N*), East (*E*) and Vertical down (*V*) direction. An input is only required for the *X*- and *Y*-axes. The *Z*-axes is then determined automatically by *GeoStress* because it is overdetermined by four given variables. Press the *Adjust* button to adjust the system. This procedure does not change the *X*-axes, but finds a *Y*-axes in the (former) *XY* plane close to the input of *Y* but perpendicular to the *X*-axes. Iterate a few times if one of the two variables (*Y*-trend or *Y*-plunge) should be fixed exactly.

GeoStressCmd: In the GeoStressCmd Command String the trend and plunge of the x and y-axis are (if required for the function) included as four comma separated values. They are identified as xtr, xpl, ytr, and ypl in the following.

4.1.2 Geo-coordinates

Enable any of the check boxes and press the *Calculate* button if the node coordinates are required in the {*N*, *E*, *V*} system. Please note that the node coordinates describe (by default) the deformed geometry of the model (see chapter "Post-processing Finite Element Data" of the Tecplot user manual) and therefore the derived geo-coordinates too.

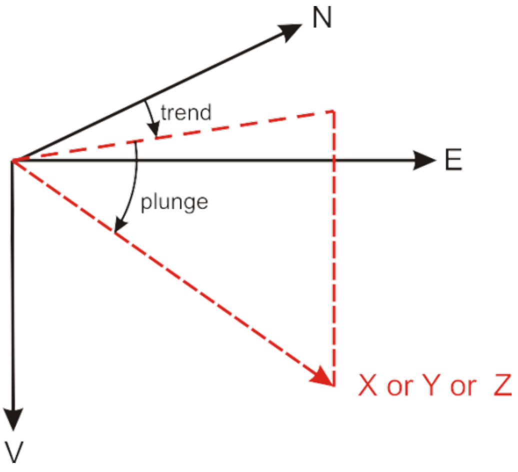


Fig. 4.1-2: Definition of the geo-orientation.

The trend is defined as the angle between the north direction and the coordinate axis, the plunge is the angle between horizontal projection of an axis and its dip.

GeoStressCmd: The computation of geo-coordinates requires general information on the orientation of axes. Each component, north (n1), east (n2) and down (n3), can be computed separately. Then the following command string syntax is used:

```
"Geo-Coordinates, xtr, xpl, ytr, ypl, n1, n2, n3"
```

A model with common axes orientations which requires only the computation of the down component results in the following Command String:

```
"Geo-Coordinates, 90, 0, 0, 0, 0, 0, 1"
```

4.1.3 Stress scaling factor and change of sign for compression

In contrast to the geosciences community the commercial finite element software packages define compressive stresses as negative. The default settings in the stress scaling factor dialog box is -1.000e-06 which changes the stress state to compression positive and it converts the units of the stress variables from Pascal (Pa) in the Abaqus **.odb*-file to Mega Pascal (MPa).

GeoStressCmd: The stress scaling factor is included in the GeoStressCmd Command String usually right behind the orientation of the axes, identified by ssf in the examples.

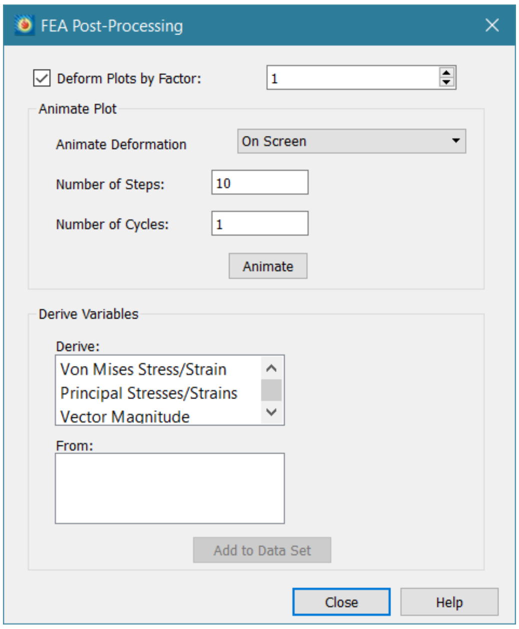
Note that units for the original six stress tensor components from the **.odb*-file (XX Stress, XY Stress, etc.) are not changed, but only the values that are derived with *GeoStress*. Furthermore, the units for the cohesion values expected in the tab cards *FP* (Fracture Potential), *CFS* (Coulomb Failure Stress) and Slip and Dilation Tendency (*ST* and *DT*) are expected in the same unit as defined in the general setting in *GeoStress*.

4.1.4 Plot scaling factor

The output of a model also includes the deformation of the geometry due to the individual displacement of the nodes. In order to exaggerated small deformations, the user can set the plot scaling factor. This is done when the *.odb file is loaded for the first time and when advanced options are chosen (see subsection 2.6). In the appearing dialog box (Fig. 4.1-3) the tick box *Deform Plots by Factor* changes the plot scaling factor; the default value of Tecplot 360 EX as well as for the Add-on *GeoStress* is 1, which means no exaggeration. This value can also be changed using the menu bar Tools/FEA Post-Processing, but once the output file is saved as a *.plt file the plot deformation factor cannot be changed anymore.

GeoStressCmd: If required, the plot scaling factor is included in the GeoStressCmd Command String usually after the stress scaling factor, identified by psf in the examples.

Note that the user has to make sure that the finally stored plot scaling factor that is used by Tecplot 360 EX is consistent with the plot scaling factor to be given in the *GeoStress* Add-on in the first page general setting (Fig. 3.1-1). If this consistency is not given, the calculation of the variable *True Vertical Depth* (*TVD*) and kinematic variables on faults would not be correct. These values are calculated for each step and therefore a variable that changes over time due to the deformation of the geometry.



☑ Deform Plots by Factor: 1

Animate Deformation: On Screen

Number of Steps: 10

Number of Cycles: 1

Fig. 4.1-3: Plot scaling factor setting in Tecplot 306 EX.

The Plot scaling factor chosen within Tecplot 360 EX must be consistent with the setting in the general page. Otherwise the calculation of relative displacement and the *True Vertical Depth* (*TVD*) is not correct.

4.1.5 True vertical depth (TVD)

The variable *TVD* is the true depth below the surface of the model geometry. It has the same units as the model (usually meters). As stated in the previous section *TVD* also considers the deformation and thus the plot scaling factor used by Tecplot 360 EX and the Add-on *GeoStress* must be consistent. The value of *TVD* is always calculated at the end of each time step; i.e. the *TVD* value of the first time-step (first *Zone*) is the one at the end of the first Abaqus increment and thus contains already deformations from the model.

To perform the calculation *GeoStress* needs an input as a **.txt*-file where the original numbering of all nodes that are located at the surface of the model are provided. The format of this input file is given in the following example.

Example:

```
** ABAQUS Input Deck Generated by HyperMesh Version 12.0.0.85
**
*NODE
100239,  681815.55544868,  268164.90825397,  407.77080517579
100247,  684577.50712194,  268650.91587525,  450.55165463753
**
```

Lines that start with **** are comments and can be inserted before or after the block of data. The key word **NODE* is compulsory and indicates the start of the read in of the data. Each data line starts with the original node number optional followed by the node coordinates. However, *GeoStress* reads in only the numbering of the nodes and not their coordinates since Tecplot 360 EX knows the link between the nodes (variable *Node UserID*) and their coordinates (variables *X, Y, Z*). Field separator can be a blank, tab stops or comma. For safety reasons, it is recommended to finalize the read in of data with a comment line.

GeoStressCmd: In GeoStressCmd general information on the axes orientation and the plot scaling factor are required in addition to the file with the nodes of the topography (fname). Please note, that the full path to the file is required. The Command String syntax is:

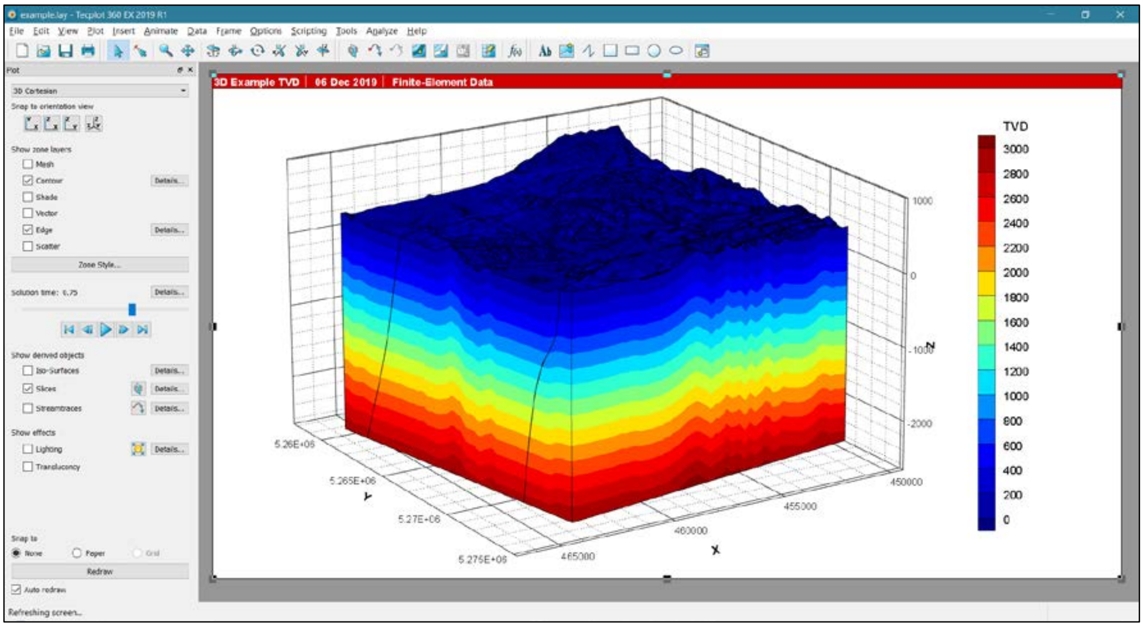
```
"TVD, xtr, xpl, ytr, ypl, psf, fn"
```

An exemplified command string is:

```
"TVD, 90.0, 0.0, 0.0, 0.0, 1.0,
C:\\Desktop\\Tecplot_Example\\topography_nodes_only.txt"
```

4.1.6 Application examples

In order to calculate the variable *TVD* the file *topography_nodes_only.txt* located in the folder Geometry has to be loaded using the button *Load topography* on the page *General* of the *GeoStress* Add-on (Fig. 4.1-1). The calculation is performed for each increment of the model considering also the changes of the *TVD* with time. For high-resolution models with many nodes at the model surface (i.e. the topography nodes) and lots of increments the *TVD* calculation can be quite time-consuming.



Plot

3D Cartesian

Show zone layers
☐ Mesh
☑ Contour
☐ Shade
☐ Vector
☑ Edge
☐ Scatter

Solution time: 0.75

Show derived objects
☐ Iso-Surfaces
☑ Slices
☐ Streamtraces

Show effects
☐ Lighting
☐ Translucency

Snap to
● None ○ Paper ○ Grid

☑ Auto redraw

Refreshing screen...

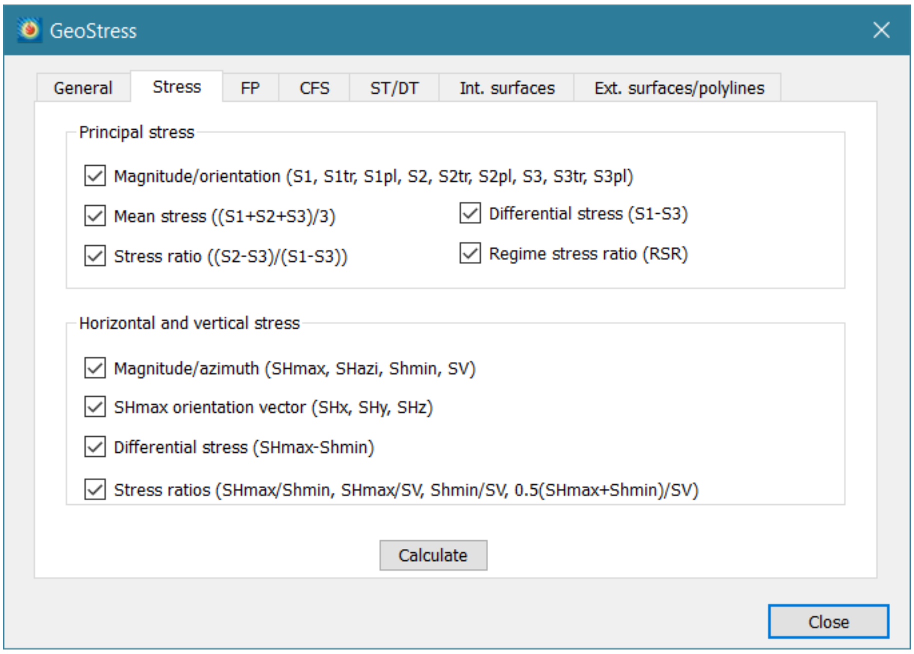
Fig. 4.1-4: Calculation of the variable True Vertical Depth *TVD*.

The plot shows the *TVD* with three times exaggeration in the vertical in increment 0.75. Note that the *TVD* is reflecting the topography.

4.2 Stress value calculations

4.2.1 Theory and background information

The page *Stress* (Fig. 4.2-1) offers the calculation of a wide range of stress variables that are derived from the six components of the 3D stress tensor. An overview as well as details and explanation of the different variables that are calculated on this page are summarized in Table 4.2-1. The page is divided in two parts. The upper area provides all information on the principal stresses *S1*, *S2* and *S3* and variables derived from these principal stresses. Furthermore, it offers to calculate from the stress tensor components the *Regime Stress Ratio* (*RSR*) that is a continuous scale for the visualization of the stress regime (Simpson, 1997). The lower part offers a range of stress variables and stress ratios for the horizontal and vertical stress components. The selected variables are calculated for the whole model volume and each time step. However, when e.g. a cross section is extracted over time (example is described in Fig. 2.2-3 and Fig. 2.2-4 in section 2.2) only the existing variables are interpolated onto these geometries after they are generated with *GeoStress* and stored in the Tecplot 360 EX plt-file format.



Principal stress

☑ Magnitude/orientation (S1, S1tr, S1pl, S2, S2tr, S2pl, S3, S3tr, S3pl)
☑ Mean stress ((S1+S2+S3)/3)
☑ Differential stress (S1-S3)
☑ Stress ratio ((S2-S3)/(S1-S3))
☑ Regime stress ratio (RSR)

Horizontal and vertical stress

☑ Magnitude/azimuth (SHmax, SHazi, Shmin, SV)
☑ SHmax orientation vector (SHx, SHy, SHz)
☑ Differential stress (SHmax-Shmin)
☑ Stress ratios (SHmax/Shmin, SHmax/SV, Shmin/SV, 0.5(SHmax+Shmin)/SV)

Fig. 4.2-1: Page *Stress*.

The page is divided in two parts. The upper area provides all variables related to the three principal stresses and the *Regime Stress Ratio* (*RSR*). The lower part provides a range of stress variables related to the horizontal and the vertical stress.

Note that new variables that are calculated with the equations option are not stored in the plt-file, but only on the lay-file. A closing and reopening of the lay-file thus requires a new calculation of the variables. Thus, it is recommended to calculate first all new variables that are needed for the whole model volume and to store these explicitly in the *.plt file and then create a new *Zone*. Furthermore, the interpolation of bi-polar variables (e.g. SHmax orientation SHazi) e.g. on slices can result in wrong results due to the ambiguity of 0° and 180°. When these two values are on either side of the plane on which the interpolation is executed the arithmetic mean would be 90° which is not correct. To avoid this we recommend using the *GeoStress* external surfaces.

GeoStressCmd: In *GeoStressCmd*, general information on the axes orientation and the stress scaling factor are required. This is followed by definition of which variables are to be computed, indicated by n1 to n9. The order is according to Table 4.2-1. The syntax of the command string is:

```
"Stress, xtr, xpl, ytr, ypl, ssf, n1, n2, n3, n4, n5, n6, n7, n8, n9"
```

exemplified by:

```
"Stress, 90.0, 0.0, 0.0, 0.0, -1.0e-6, 0, 1, 0, 1, 0, 1, 0, 0, 0"
```

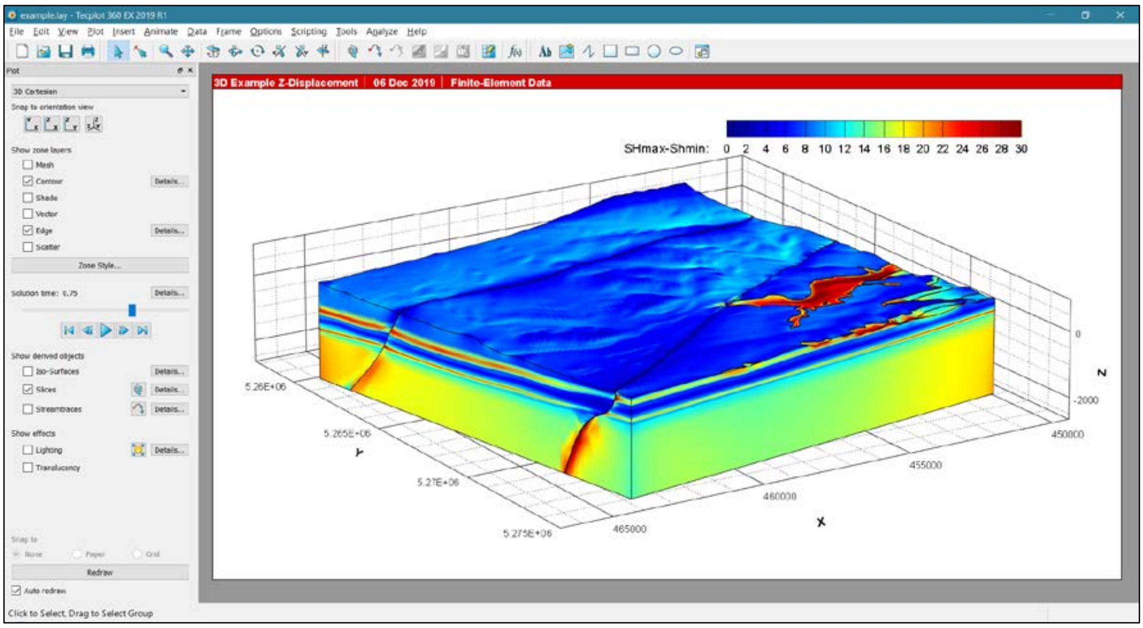
Tab. 4.2-1: Overview of stress variable calculations with *GeoStress* page *Stress*.

The first column shows the variable as it appears in the variable list in Tecplot 360 EX. The second column gives an explanation or background information.

Variables	**Definition/Comment**
S1, S2, S3 *S1tr, S1pl* *S2tr, S2pl* *S3tr, S3pl*	Magnitudes of the three principal stresses (the stress unit depends on the choice in the general settings of the first page of *GeoStress*). Orientation in terms of trend (tr) and plunge (pl) in degrees of the three principal stresses in the System {*N, E, V*} as defined in the general settings of the first page of GeoStress (see Fig. 4.1.-2).
(S1+S2+S3)/3	Mean stress. Note that *(S1+S2+S3)/3 = (Sxx+Syy+Szz)/3* since this is one of the three invariants of the stress tensor.
(S2-S3)/(S1-S3)	Stress ratio. When *S1-S3*=0 the stress state is lithostatic (sometimes also called isotropic) and *(S2-S3)/(S1-S3)*=1.
S1-S3	Differential stress as the difference between the largest (*S1*) and the smallest principal stress (*S3*).
RSR	The Regime Stress Ratio (*RSR*) is calculated after the formula given in Simpson (1997) and provides a continuous scale for the stress regime. The *RSR* is basically a combination of the Andersonian stress regime index n (Anderson, 1905; 1951) and the stress ratio *SR* $RSR = (n + 0.5) + (-1)^n (SR - 0.5) \; = \begin{cases} n = 1 \; if \; Shmin \le SV \le SHmax \\ n = 2 \; if \; SV \le Shmin \le SHmax \\ n = 0 \; otherwise \end{cases}$ where *SR=(S2-S3)/(S1-S3)*. The *RSR* variable ranges from radial extension (*RSR*=0) over normal faulting (*RSR*=0.5), strike-slip (*RSR*=1.5) and thrust faulting (*RSR*=2.5) to constriction (*RSR*=3), including the transitional stress regimes of transtension (*RSR*=1) and transpression (*RSR*=2).
SHmax, SHazi *Shmin, SV*	Magnitude of the vertical stress SV and maximum and minimum horizontal stress *SHmax* and *Shmin*, respectively. *SHazi* is the orientation of *SHmax* in degrees from North and lies between 0-180° due to the bipolar character of stress orientations.
SHx, SHy, SHz	The three components of the of the *SHmax* vector. Note that *SHz* should be zero and that only the two components *SHx* and *SHy* are needed to plot with Tecplot 360 EX the S_{Hmax} orientation as a vector. Note that these components are given in the old coordinate system to allow Tecplot 360 EX to plot the vector.
SHmax-Shmin	Horizontal differential stress.
SHmax/Shmin *SHmax/SV* *Shmin/SV* *0.5(SHmax+Shmin)/SV*	Various stress ratios. If *Shmin*=0 the ratio *SHmax/Shmin* is arbitrarily set to -999, if *SV*=0 the other three ratios are set to -999.

4.2.2 Application examples

The following two figures show the results of the horizontal stress difference S_{Hmax}-S_{hmin} (Fig. 4.2-2) and the Regime Stress Ratio RSR (Fig. 4.2-3), respectively.



3D Cartesian

☐ Mesh
☑ Contour
☐ Shade
☐ Vector
☑ Edge
☐ Scatter

Solution time: 0.75

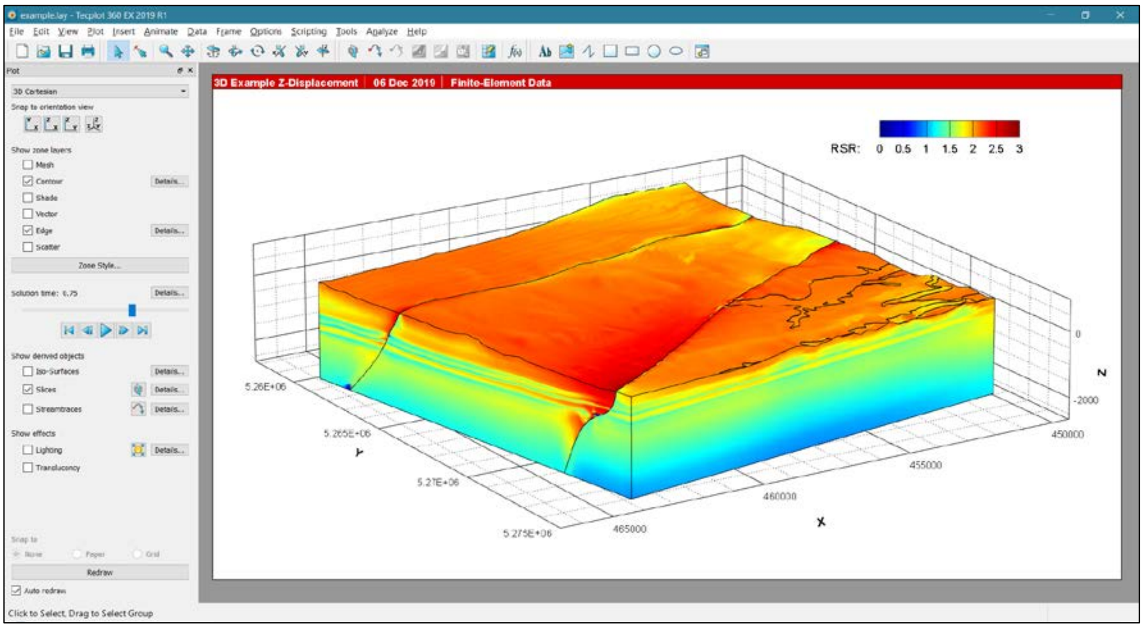
☐ Iso-Surfaces
☑ Slices
☐ Streamtraces

☐ Lighting
☐ Translucency

☑ Auto redraw

Click to Select, Drag to Select Group

Fig. 4.2-2: Contour plot of the horizontal stress difference $S_{Hmax} – S_{hmin}$.



3D Cartesian

☐ Mesh
☑ Contour
☐ Shade
☐ Vector
☑ Edge
☐ Scatter

Solution time: 0.75

☐ Iso-Surfaces
☑ Slices
☐ Streamtraces

☐ Lighting
☐ Translucency

☑ Auto redraw

Click to Select, Drag to Select Group

Fig. 4.2-3: Contour plot of the Regime Stress Ratio (RSR).

4.3 Fracture potential

4.3.1 Theory and background information

The Fracture Potential (*FP*) is described by Connolly and Cosgrove (1999) and expresses the ratio of the actual maximum shear stress (S1-S3)/2 in the model and the critical value of acceptable shear stress from an empirical failure criterion. It thus quantifies how far the actual stress state is away from failure, i.e. the generation of fractures in a rock mass subjected to stress. This definition holds only for shear failure which is considered here since the crust is generally in a compressional state of stress.

The critical differential stress used in *GeoStress* is based on the Navier-Coulomb criterion and hence dependent on the internal coefficient of friction μ_i of the rock and the cohesion *C*. These two parameters have to be specified in the *GeoStress* settings in order to calculate the *FP* for a given stress state. The units of the values have to be in agreement with the units given in the *GeoStress* page *General setting* (Fig. 4.1-1). In particular *FP* is a relative measure for the Coulomb shear failure criterion in a volume with a given stress state:

$$FP = \frac{actual\ maximum\ shear\ stress}{acceptable\ shear\ stress}$$

It is calculated on the optimal oriented plane and it is defined as the ratio with the following definitions of the actual and the acceptable maximum shear stress:

$$actual\ maximum\ shear\ stress = \tfrac{1}{2}(S1 - S3)$$

$$acceptable\ maximum\ shear\ stress = \left(\frac{C}{\tan\phi} + \tfrac{1}{2}(S1 + S3)\right)\sin\phi = C\cos\phi + \tfrac{1}{2}(S1 + S3)\sin\phi$$

C is the cohesion, ϕ the friction angle ($tan\phi = \mu_i$) and *S1* and *S3* are the largest and smallest principal stress, respectively.

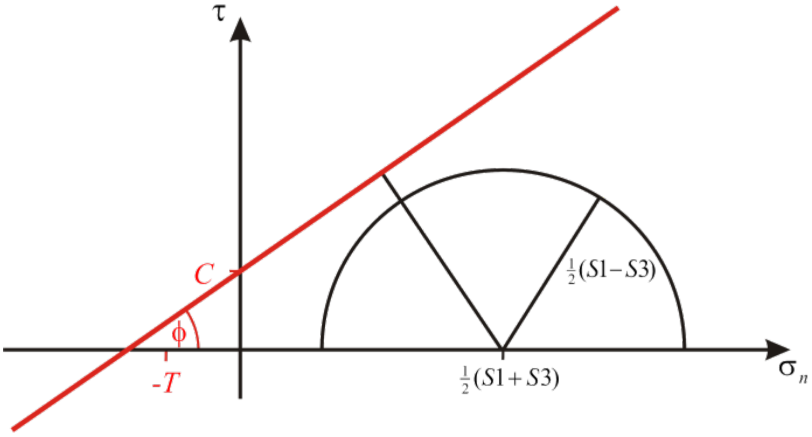


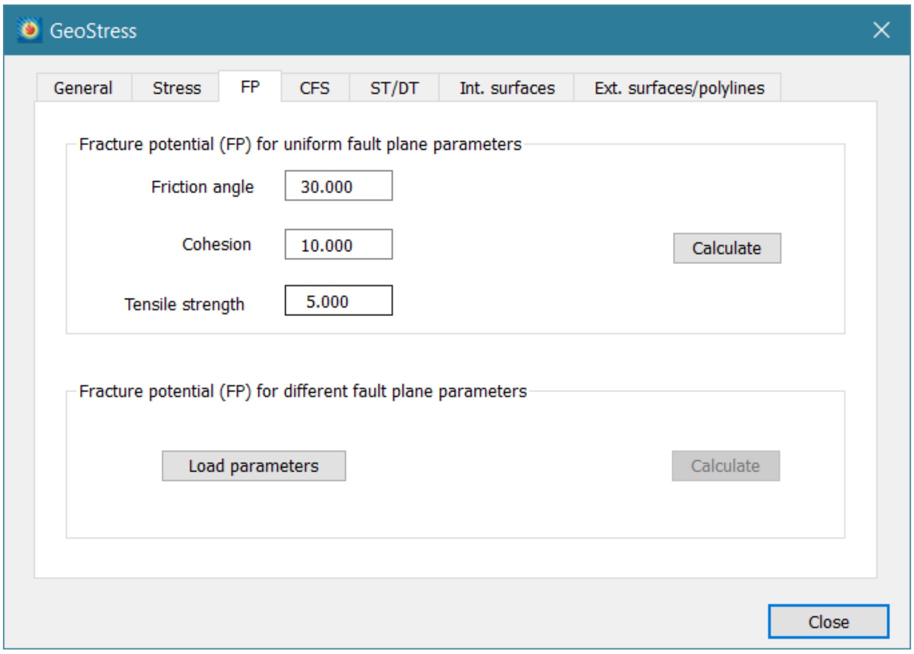
Fig. 4.3-1: Coulomb failure in a 2D plane.

C is the cohesion, T the tensional strength, ϕ the friction angle, τ is the shear stress and σ_n the normal stress of the optimal oriented plane.

Under specific circumstances the calculation of the *FP* can result incorrect values. E.g. when the differential stress *S1-S3* is very small and *S3* negative, the fault would not fail in shear (Mode II), but in opening/dilation (Mode I) due to the tensional stresses (and the tensional strength). To account for these special cases the following rule is applied:

$$FP = \left\{ \begin{array}{lll} -999 & \text{if} \quad \frac{1}{2}(S1+S3) \le -T & \text{(tensile fracture)} \\ -9999 & \text{if} \quad -T < \frac{1}{2}(S1+S3) \le \dfrac{C}{\tan\phi} & \text{(undefined)} \\ & \frac{1}{2}(S1-S3)/(C\cos\phi + \frac{1}{2}(S1+S3)\sin\phi) & \text{otherwise} \end{array} \right.$$

where *T* is the tensional strength of the rock. This value can be set in the *GeoStress* page for the *FP* calculations (Fig. 4.3-2). Again, the user of the Add-on is responsible that the used units are consistent with the stress scaling factor.



Fracture potential (FP) for uniform fault plane parameters

Friction angle 30.000

Cohesion 10.000

Tensile strength 5.000

Fracture potential (FP) for different fault plane parameters

Fig. 4.3-2: Page *Fracture Potential*.

The FP can be calculated either with homogeneous rock properties (upper section) or individual values for each rock type in the model (lower section).

The *FP* values can be calculated either with homogeneous rock properties for the whole model (upper section of the *FP* page) or with individual values for each rock material of the model (lower section of the *FP* page). The individual material values (*C*, ϕ and *T*) for a node are calculated as the mean value from all elements that share this node. The material definition of the elements is given in the variable *Material ID* and is provided by Abaqus and thus also given as a variable in Tecplot 360 EX. The assignment of the variable *Material ID* to the material properties is done with a text file (**.txt*). This file is uploaded with the button *Load Parameter* in the graphical user interface.

Example file FP_plastic_properties.txt:

```
*****************************************************************
** Values for plasticity (friction angle (2nd column), cohesion
** (3rd column in MPa), tensile strength (4th column in MPa) for
** model NL. First column is the internal numbering of the material
** ID from Tecplot 360 EX. However, the numbering is not related
** to the order of the material definitions in the Abaqus input file.
*****************************************************************
*FP
6 38 10 5           Molasse and Quaternary Cover
8 50 20 5           Upper Malm
1 40 8 5            Wildegg Formation
7 30 8 5            Upper Dogger
10 23 4 5           Opalinus Clay
3 30 8 5            Lias and Upper Middle Keuper
2 34 28 5           Gipskeuper
9 45 23 5           Upper Muschelkalk
5 40 20 5           Middle and Lower Muschelkalk
4 40 30 5           Pre-Mesozoic Basement
*****************************************************************
```

The key word **FP* is compulsory right before the data lines. Comments are indicated with **; the end of data to read in has to be denoted with a comment **. Column 1 defines the *Material ID*, column 2 the *friction angle ϕ*, column 3 the *cohesion C* and column 4 the *tensional strength T*. Each line can be completed by an optional comment. The field separator can be a blank, comma or tab stop.

GeoStressCmd: In *GeoStressCmd*, general information on the stress scaling factor are required in the command string. In case of uniform fracture parameters this is followed by a 0 at the position of n1, the friction angle (fr), the cohesion (co), and the tensile strength (te) of the rock. This command string syntax is:

```
"FP, ssf, n1, fr, co, te"
```

exemplified by:

```
"FP, -1.0e-6, 0, 30.0, 10.0, 5.0"
```

In case of different parameters, the command string contains a 1 at the position of n1 and the file (full path) with the fracture parameters. Then, the command string syntax is:

```
"FP, -1.0e-6, 1, C:\\Desktop\\Tecplot_Example\\FP_plastic_properties.txt"
```

4.3.2 Application examples

Fig. 4.3-3 shows the calculation of the Fracture Potential with one cohesion and friction angle for the whole model volume. Fig. 4.3-4 displays the material ID number that is needed to create the file that connects the *Material ID* with variable cohesions and friction angles for each individual geomechanical unit of the model. Then Fig. 4.3-5 shows the resulting *FP* values.

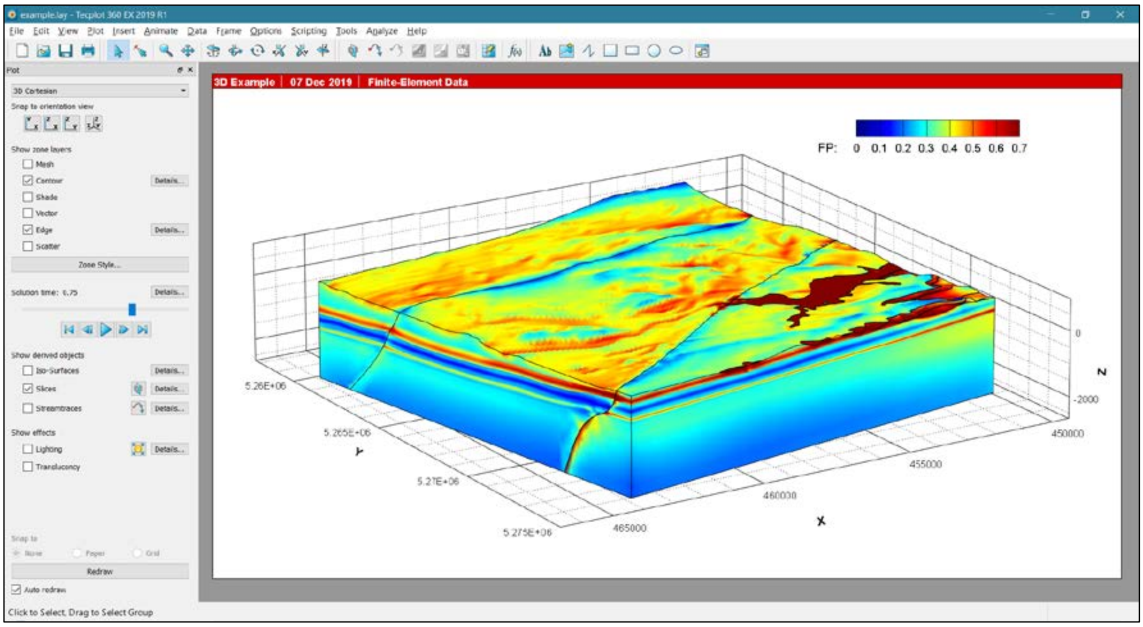


Fig. 4.3-3: Calculation of the Fracture Potential with homogeneous rock properties.

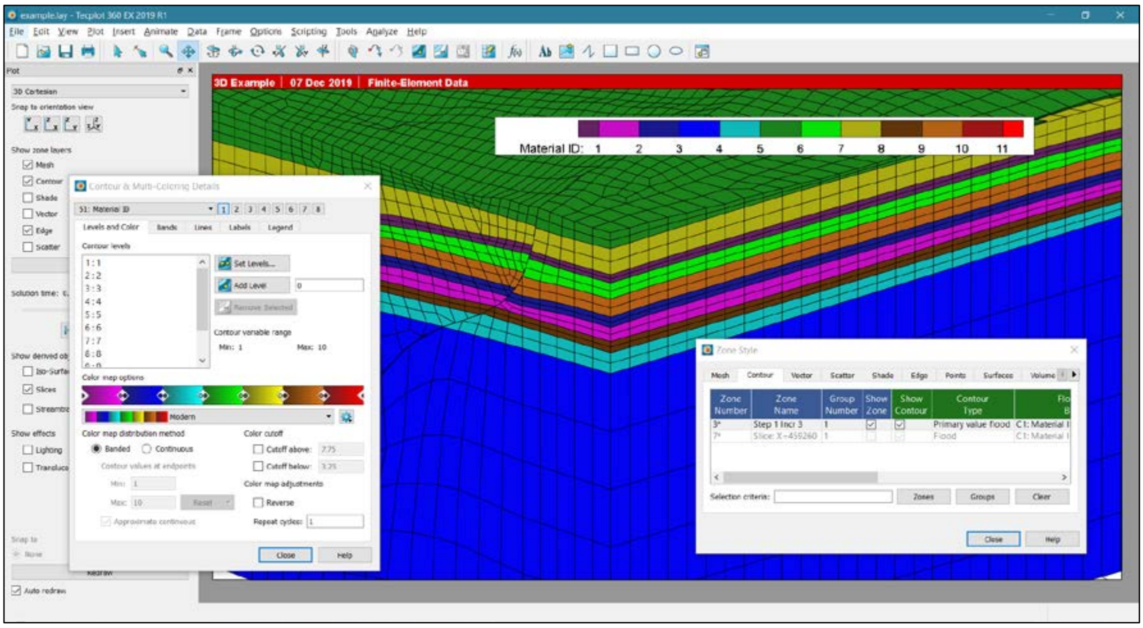
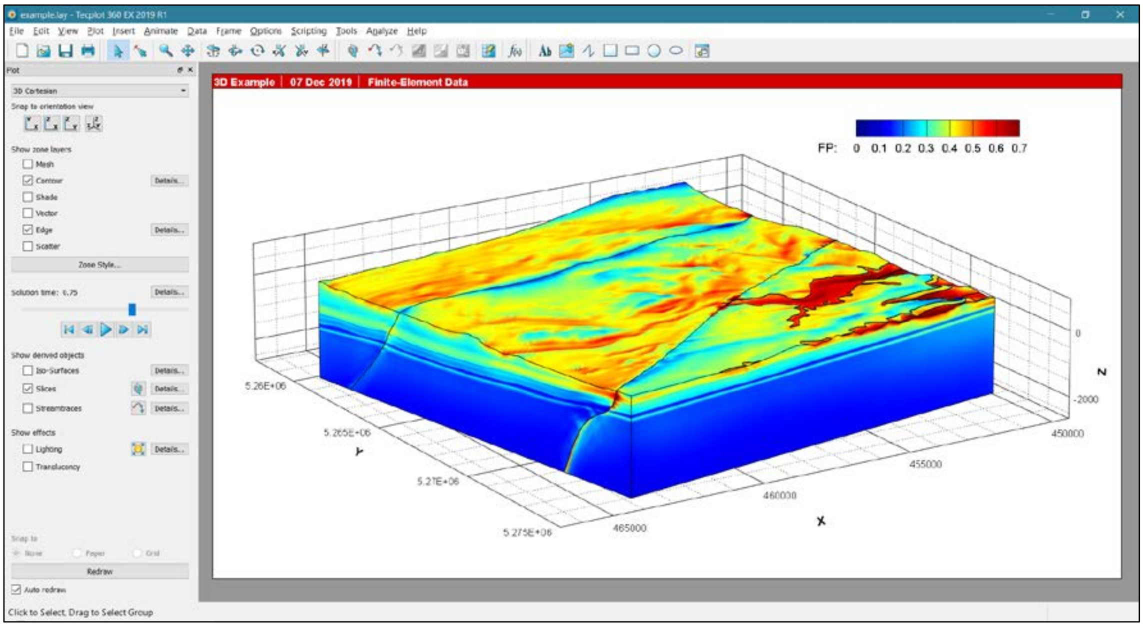


Fig. 4.3-4: Visualization of the Material ID number.

To get the correct Fracture Potential values a file that links the *Material ID* number, cohesion and friction angle for each unit must be provided. Use the Primary value flood to display for each unit the number correctly.



3D Cartesian

☐ Mesh
☑ Contour
☐ Shade
☐ Vector
☑ Edge
☐ Scatter

Solution time: 6.75

☐ Iso-Surfaces
☑ Slices
☐ Streamtraces

☐ Lighting
☐ Translucency

☑ Auto redraw

Click to Select, Drag to Select Group

Fig. 4.3-5: Plot of the Fracture Potential with inhomogeneous rock properties.

Note the difference to the results displayed in Fig. 4.3-3 where a homogeneous rock property values for the cohesion and the friction angle is used.

4.4 Coulomb failure stress

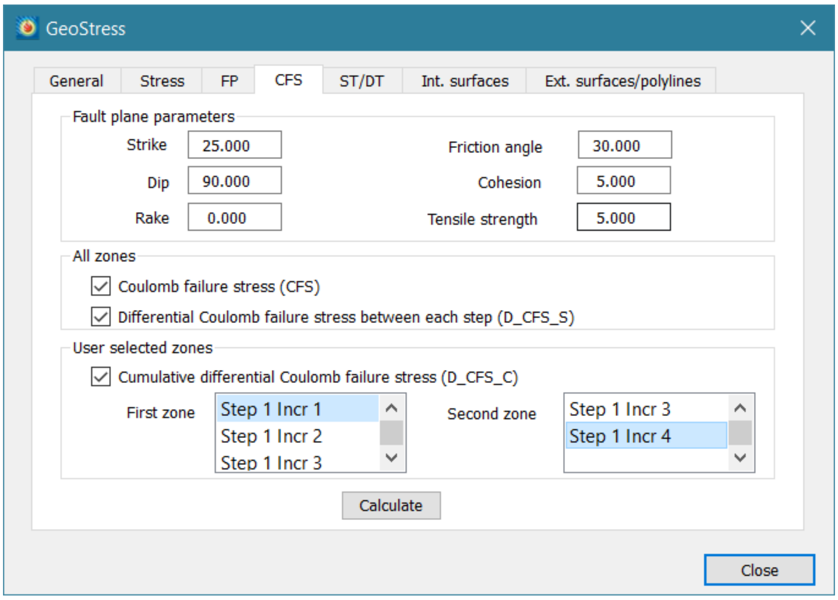
4.4.1 Theory and background information

The Coulomb failure stress (*CFS)* for a given fault or segment of a fault is the difference between the shear stress τ and the Coulomb stress $\tan(\phi)\sigma_n - C$

$$CFS = \tau - \tan(\phi)\,\sigma_n - C$$

where τ is the shear stress on a plane that represents a fault or a segment of a fault, *tan(ϕ)* is the static friction coefficient μ expressed with the tangent of the friction angle ϕ, σ_n the normal stress and *C* the cohesion. The shear stress is defined on the fault given by its strike and dip in a *a-priori* selected rake direction; the latter is essential to assign the correct sign of the shear stress on the fault. The Coulomb failure stress is the resistance to this shear stress. Thus, negative values quantify how far the actual stress state is away from failure. Values close to zero would indicate that the fault is critically stressed and positive values would indicate that the fault would slip to release the excess of shear stress. Note that like slip tendency CFS provides only valid information about potential failure on one specific plane. Furthermore, in contrary to slip tendency CFS is also dependent on the rake vector. Hence its significance is limited to a specific focal mechanism on a specific plane.

However, the commonly used definition of CFS does not distinguish between compression and tension and thus, when normal stresses are sufficiently negative faults would not fail in shear (Mode II) but in dilation (opening, Mode I). Thus, when the normal stress σ_n is a tension (negative sign) and exceeds the given tensile strength *T* ($-\sigma_n \geq T$) *GeoStress* assigns CFS=-999.



Fault plane parameters

Strike 25.000

Dip 90.000

Rake 0.000

Friction angle 30.000

Cohesion 5.000

Tensile strength 5.000

All zones

☑ Coulomb failure stress (CFS)

☑ Differential Coulomb failure stress between each step (D_CFS_S)

User selected zones

☑ Cumulative differential Coulomb failure stress (D_CFS_C)

First zone: Step 1 Incr 1
Step 1 Incr 2
Step 1 Incr 3

Second zone: Step 1 Incr 3
Step 1 Incr 4

Fig. 4.4-1: Page *Coulomb Failure Stress*.

The page is divided into three sections. In the upper section the orientation of fault plane parameter is stated for which the Coulomb failure stress is calculated and the properties of the fault plane. In the second section the user can select to calculate the total *CFS* and the differential *CFS* (*D_CFS_S*) between all *Zones*. In the lower section the user select two *Zones*. Between these the cumulative differential *CFS* (*D_CFS_C*) is estimated.

Fault plane parameters

GeoStress defines strike, dip and rake following the conventions by Aki and Richards (1980). Fault strike is the direction of a line created by the intersection of a fault plane and a horizontal surface. It can range from 0° to 360° relative to North. Strike is always defined such that a fault dips to the right side of the trace when moving along the trace in the strike direction. Thus, the hanging-wall block of a fault is always to the right, and the footwall block to the left. This is important because the rake which defines the (potential) slip direction is defined as the movement of the hanging wall relative to the footwall block. The fault dip is the angle between the fault and a horizontal plane and ranges from 0° for a horizontal fault to 90° for a vertical fault. Rake is the direction a hanging wall block moves during rupture, as measured on the plane of the fault. It is measured relative to fault strike, ±180°.

For an observer standing on a fault and looking in the strike direction, a rake of 0° means that the hanging wall (or the right side of a vertical fault) moves away from the observer in the strike direction (left lateral motion). A rake of ± 180° means the hanging wall moves toward the observer (right lateral motion). For any rake > 0° and < 180°, the hanging wall moves up, indicating reverse motion on the fault; for any rake < 0° and > -180° the hanging wall moves down, indicating normal motion on the fault.

Calculation of different CFS values

The calculation of *CFS* values is divided into two sections *All Zones* and *User selected Zones*. The Coulomb failure stress *CFS* and the single difference in Coulomb failure stress (*D_CFS_S*) can be calculated for all *Zones* of the 3D model (Fig. 4.4-1). The *D_CFS_S* is the difference of *CFS* between successive *Zones* (model steps) and it is assigned to the latter *Zone*. Thus

D_CFS_S is zero for the first *Zone* and has a value for all other *Zone*s. *D_CFS_S* is flagged by -999 for a node if at least one of the single *CFS* values of the node is flagged by -999.

The lower section allows the user to define between which *Zone*s (model steps/increments) the cumulative differential Coulomb failure stress (*D_CFS_C*) is calculated. The *D_CFS_C* variable is zero for the selected First *Zone* and then changing from step to step until the Second *Zone* is reached. Positive differential *CFS* values (single and cumulative) indicate that the stress state has changed from one step to the next towards failure, i.e. the fault (with the given orientation, dip and rake) has been brought closer to failure. Vice versa negative differential *CFS* values indicate that the fault has been brought further away from failure.

GeoStressCmd: In *GeoStressCmd*, general information on the axes orientation and the stress scaling factor are required. This is followed by definition of the fault strike (fst), dip (fdp), rake (frk), as well as the friction angle (fr), the cohesion (co), and the tensile strength (te) of the rock. Then, the computed variables are specified. CFS (n1), D_CFS_S (n2), and D_CFS_C (n3). If n3=0, the command string syntax is:

```
"CFS, xtr, xpl, ytr, ypl, ssf, fst, fdp, frk, fr, co, te, n1, n2, n3"
```

exemplified by:

```
"CFS, 90.0, 0.0, 0.0, 0.0, -1.0e-6, 25.0, 90.0, 0.0, 30.0, 5.0, 5.0, 1, 1, 0"
```

However, if the cumulative differential stress (D_CFS_C) should be computed, n3=1. Then, the two zones that should be compared need to be included. Please note, that the zone name is case-sensitive. The command string syntax is then:

```
"CFS, xtr, xpl, ytr, ypl, ssf, fst, fdp, frk, fr, co, te, n1, n2, n3 , zn1,
zn2"
```

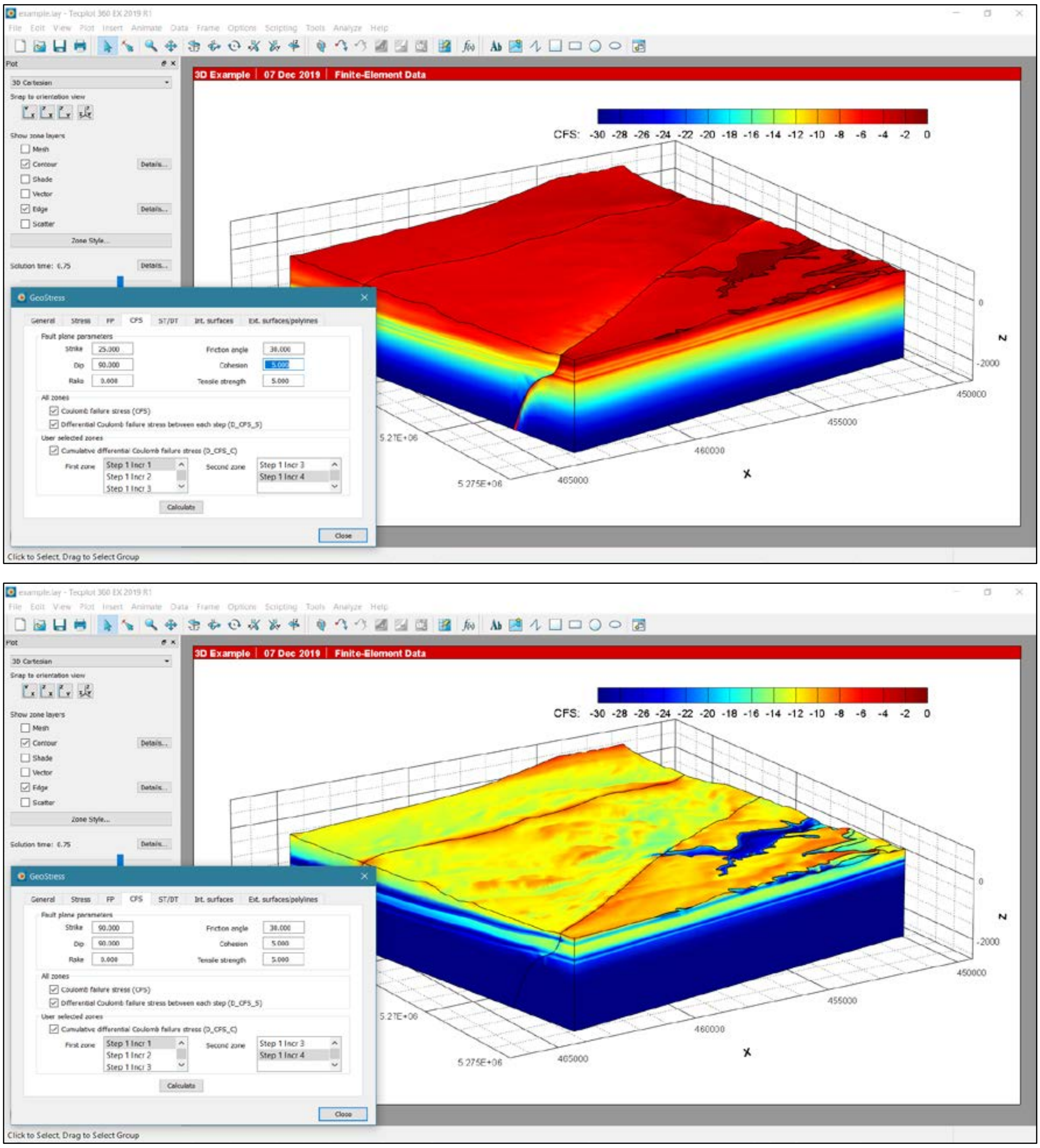
exemplified by:

```
"CFS, 90.0, 0.0, 0.0, 0.0, -1.0e-6, 25.0, 90.0, 0.0, 30.0, 5.0, 5.0, 1, 1, 1,
Step 1 Incr 2, Step 1 Incr 4"
```

4.4.2 Application examples

Figure 4.4-2 shows the Coulomb failure stress *CFS* in MPa as a contour plot for the model volume. The *CFS* values in the upper figure are calculated for vertical faults (dip=90°) with a strike of 25°N and a rake direction of 0°. This orientation is almost optimal oriented in the given stress field. In the lower figure the *CFS* values are calculated for vertical faults as well, but with a strike of 90°N. Here the resulting values are significantly lower since for E-W striking faults the S_{Hmax} orientation is almost perpendicular to the fault which increases the fault normal stress and decreases the shear stress on the fault in comparison to the first calculation in the upper figure. However, the interpretation of the resulting colour distribution has to be done with care. E.g. the higher *CFS* values near the fault that are implemented in the model do not indicate that the fault is closer or further away from failure. The calculation of the *CFS* values in this example has nothing to do with the strike and dip of the fault that is implemented in the model. Figure 4.4-3 shows the differential Coulomb failure stress *D_CFS_S* between successive steps. Since the displacement boundary conditions between two steps are similar the two figures look very similar except local stress changes due to the fault reactivation. Figure 4.4-4 shows the cumulative differential Coulomb failure stress *D_CFS_C* between two user selected *Zone*s. The *D_CFS_C* values are increasing from one step to the next. The values mostly positive and thus indicate that vertical sinistral strike-slip faults that strike 25°N are brought closer to failure.

Note that if you change the geometry of the fault plane (strike, dip, rake) or the fault properties (friction angle, cohesion, tensile strength) in the upper section and re-calculate any of the offered three *CFS* values (*CFS, D_CFS_S*, *D_CFS_C*) by pressing calculate button, the results in all frames are changing on the fly. This enables you to perform a fast and efficient analysis of your models results. Note that the fault parameter setting in this page does not affect the calculations of the other *GeoStress* tab cards (and vice versa).

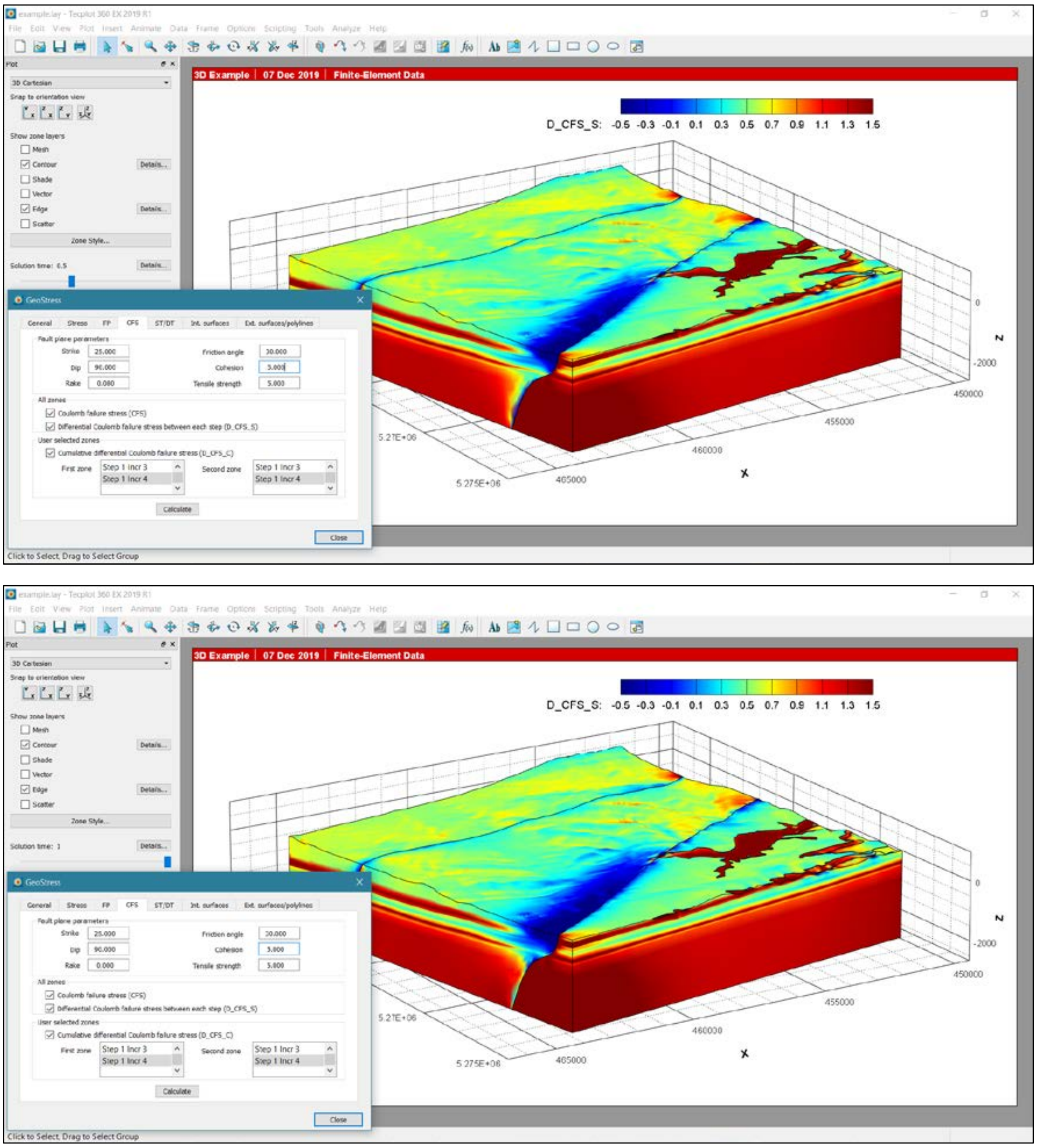


Fault plane parameters
Strike 25.000
Dip 90.000
Rake 0.000
Friction angle 30.000
Cohesion 5.000
Tensile strength 5.000
All zones
☑ Coulomb failure stress (CFS)
☑ Differential Coulomb failure stress between each step (D_CFS_S)
User selected zones
☑ Cumulative differential Coulomb failure stress (D_CFS_C)
First zone: Step 1 Incr 1
Second zone: Step 1 Incr 4

Fault plane parameters
Strike 90.000
Dip 90.000
Rake 0.000
Friction angle 30.000
Cohesion 5.000
Tensile strength 5.000
All zones
☑ Coulomb failure stress (CFS)
☑ Differential Coulomb failure stress between each step (D_CFS_S)
User selected zones
☑ Cumulative differential Coulomb failure stress (D_CFS_C)
First zone: Step 1 Incr 1
Second zone: Step 1 Incr 4

Fig. 4.4-2: Coulomb failure stress calculation.

The upper figure shows the absolute *CFS* value in MPa calculated for vertical faults (dip=90°) that strike 25°N. In the lower figure the *CFS* is calculated with the same values, but the strike has changed to 90°. In both cases the rake direction is 0°



Solution time: 0.5

Fault plane parameters
Strike 25.000
Dip 90.000
Rake 0.000
Friction angle 30.000
Cohesion 5.000
Tensile strength 5.000

All zones
☑ Coulomb failure stress (CFS)
☑ Differential Coulomb failure stress between each step (D_CFS_S)

User selected zones
☑ Cumulative differential Coulomb failure stress (D_CFS_C)
First zone: Step 1 Incr 3, Step 1 Incr 4
Second zone: Step 1 Incr 3, Step 1 Incr 4

Solution time: 1

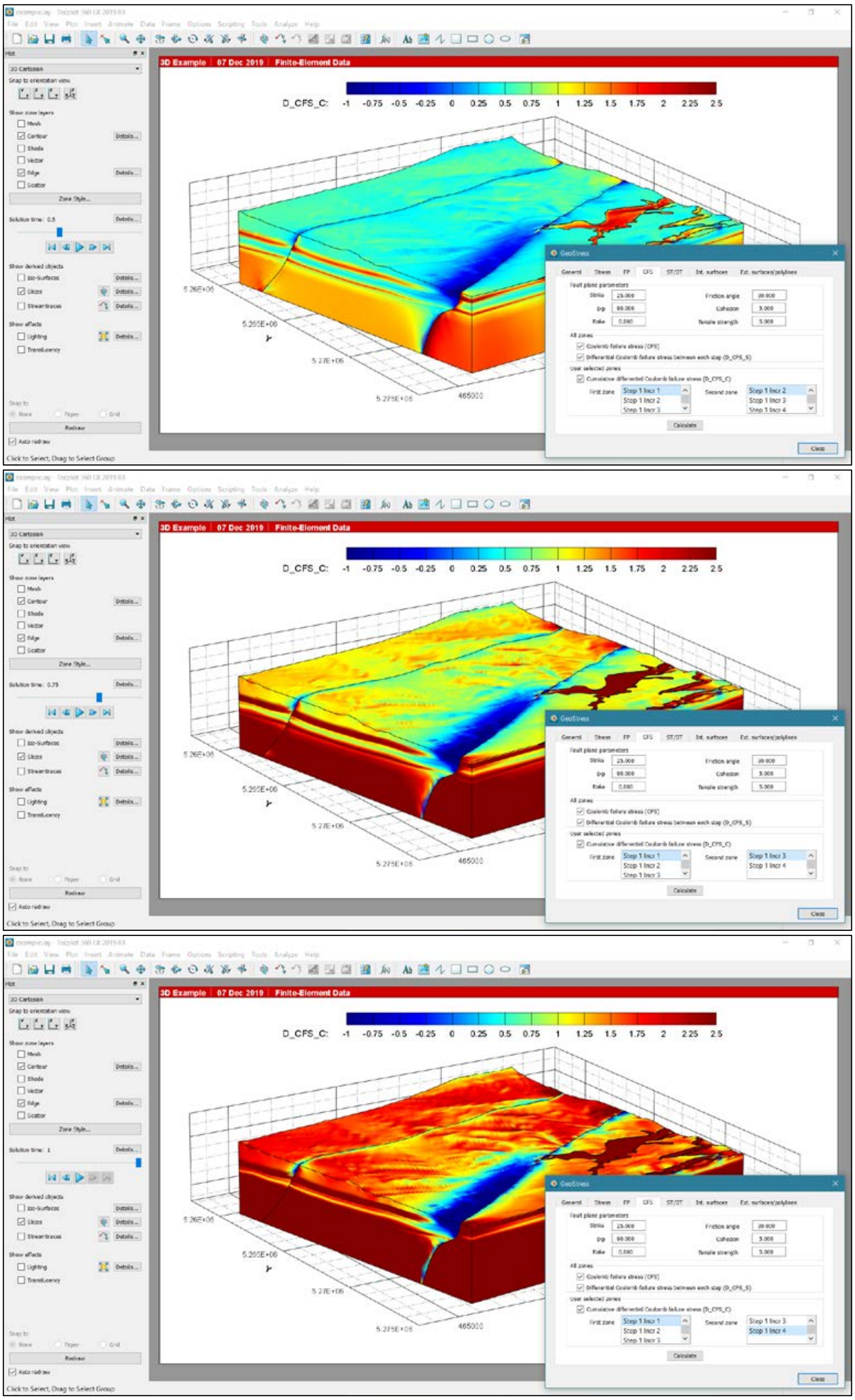
Fault plane parameters
Strike 25.000
Dip 90.000
Rake 0.000
Friction angle 30.000
Cohesion 5.000
Tensile strength 5.000

All zones
☑ Coulomb failure stress (CFS)
☑ Differential Coulomb failure stress between each step (D_CFS_S)

User selected zones
☑ Cumulative differential Coulomb failure stress (D_CFS_C)
First zone: Step 1 Incr 3, Step 1 Incr 4
Second zone: Step 1 Incr 3, Step 1 Incr 4

Fig. 4.4-3: Single differential Coulomb failure stress between all model steps.

The upper figure shows the *D_CFS_S* values in MPa for Step 1 Incr2 - Step1 Incr1. The lower figure shows *D_CFS_S* in MPa for Step 1 Incr4 - Step1 Incr4. The value of *D_CFS_S* that is presented in a *Zone* is always the difference to the previous *Zone* and it is written in the second *Zone*.



Fault plane parameters
Strike 25.000
Dip 90.000
Rake 0.000
Friction angle 30.000
Cohesion 5.000
Tensile strength 5.000

All zones
☑ Coulomb failure stress (CFS)
☑ Differential Coulomb failure stress between each step (D_CFS_S)

User selected zones
☑ Cumulative differential Coulomb failure stress (D_CFS_C)

First zone: Step 1 Incr 1 — Second zone: Step 1 Incr 2

First zone: Step 1 Incr 1 — Second zone: Step 1 Incr 3

First zone: Step 1 Incr 1 — Second zone: Step 1 Incr 4

Fig. 4.4-4: Cumulative differential Coulomb failure stress between model steps.

D_CFS_C in MPa for three steps: Upper figure is *D_CFS_C* for Step1 Incr2 – Step 1 Incr1, middle is Step1 Incr3 - Step 1 Incr2 and lower is Step1 Incr4 - Step 1 Incr3. Value of *D_CFS_C* presented in a *Zone* is always the difference to the selected first *Zone*.

4.5 Slip and dilation tendency

4.5.1 Theory and background information

The slip tendency *ST* value is a measure for the relative tendency of a pre-existing fault to reach re-activation by shear failure (Mode I) in the given 3D stress condition. *ST* on a fault is defined as the ratio of the maximum shear stress τ_{max} to normal stress σ_n normalised by the internal coefficient of friction μ that is expressed as tan(ϕ) where ϕ is the friction angle (Morris et al., 1996). Taking also into account the cohesion *C* the *ST* value can be defined as:

$$ST = \frac{\frac{\tau_{max} - C}{\sigma_n}}{\tan \phi}$$

A *ST* value close to zero indicates stable faults, whereas *ST* values close to one refer to critically stressed faults that are mature for re-activation. For $\sigma_n \leq 0$ we assign *ST* = -999 for the same reasons stated in the previous section. When compatibility to Neves et al. (2009) is wanted, the user must set $\phi = 45°$ and *C=0*.

The dilatation tendency *DT* is a measure for the relative tendency of a pre-existing fault to dilate based on the 3D stress condition (Jaeger et al., 2015). It is defined as:

$$DT = \frac{S1 - \sigma_n}{S1 - S3}$$

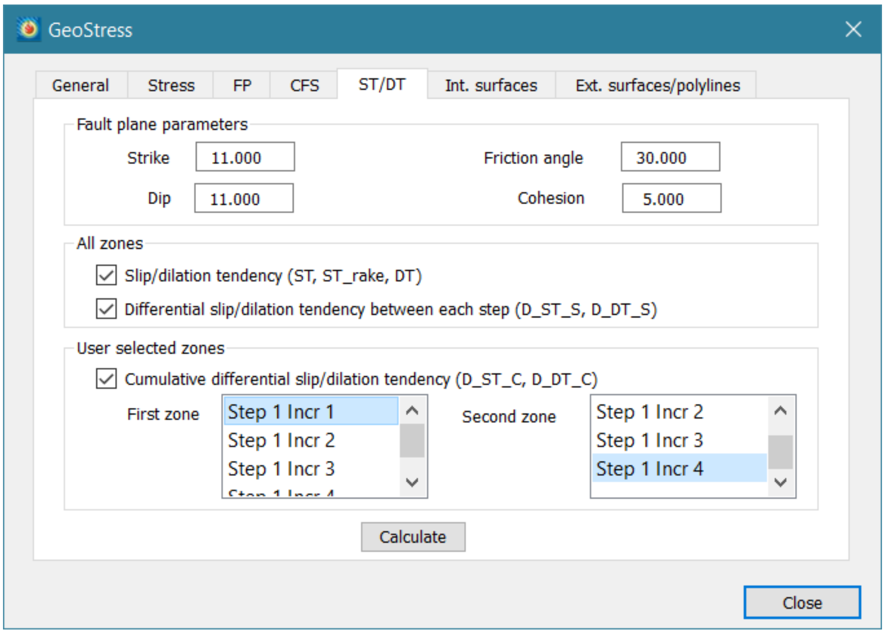
For *S1-S3* = 0 the stress state is lithostatic, i.e. the magnitudes of the three principal stresses are equal and *DT* = 1. For the fault plane parameter, the same convention holds on as explained in the previous section 4.4. The only difference is that the rake information is not needed and neither the value of the tensile strength.

4.5.2 Calculation of ST and DT values

The calculation of *ST* and *DT* is divided into the sections *All Zones* and *User selected Zones* (Fig. 4.5-1). In the first section the variables are estimated for all *Zone*s (model increments/steps). Besides the absolute values for *ST* and *DT* the differential value between consecutive *Zone*s can be estimated. The resulting variables *D_ST_S* and *D_DT_S* are written in the second *Zone*, i.e. in the first *Zone* of the model these values are zero. The variable *D_ST_S* or *D_DT_S* are flagged by -999 for a node if at least one of the single *ST/DT* values of the node is flagged. Furthermore, the value of the rake of τ_{max} (*ST_rake)* of the potential slip on the failure plane is determined.

The lower section allows the user to define between which *Zone*s (model steps/increments) the cumulative differential slip or dilation tendency (*D_ST_C, D_DT_C*) is calculated. The *D_ST_C* and *D_DT_C* variables are zero for the selected first *Zone* and then changing from step to step until the second *Zone* is reached. Positive differential *ST* and *DT* values (single and cumulative) indicate that the stress state has changed from one step to the next and that the fault is relatively closer to failure. Vice versa negative differential *ST* and *DT* values (single and cumulative) indicate that the fault has been brought relatively further away from failure.

> **Note** that the *ST* and *DT* values are relative measures and that even an increase in *ST* and *DT* can occur when the absolute distance to the failure envelope has increased.



Fault plane parameters

Strike 11.000

Dip 11.000

Friction angle 30.000

Cohesion 5.000

All zones

☑ Slip/dilation tendency (ST, ST_rake, DT)

☑ Differential slip/dilation tendency between each step (D_ST_S, D_DT_S)

User selected zones

☑ Cumulative differential slip/dilation tendency (D_ST_C, D_DT_C)

First zone
Step 1 Incr 1
Step 1 Incr 2
Step 1 Incr 3

Second zone
Step 1 Incr 2
Step 1 Incr 3
Step 1 Incr 4

Fig. 4.5-1: Page *Slip and* Dilation *tendency*.

The page is divided in three sections. In the upper section the properties and orientation of the fault plane are stated for which the slip- and dilation tendency (*ST* and *DT*) is calculated. In the middle section slip and dilation tendency are calculated as well as the rake of the potential slip (*ST_rake*) and the difference of slip and dilation tendency between each model step (*D_ST_S* and *D_DT_S*). In the lower section the cumulative difference of these values (*D_ST_C* and *D_DT_C*) between two *Zone*s selected by the user is calculated.

GeoStressCmd: In *GeoStressCmd*, general information on the axes orientation and the stress scaling factor are required. This is followed by definition of the fault strike (fst), dip (fdp), as well as the friction angle (fr) and the cohesion (co). Then the computed variables are specified. ST, ST_rake and DT (n1), D_ST_S and D_DT_S (n2), and D_ST_C and D_DT_C (n3). If n3=0, the command string syntax is:

```
"SDT, xtr, xpl, ytr, ypl, ssf, fst, fdp, fr, co, n1, n2, n3"
```

exemplified by:

```
"SDT, 90.0, 0.0, 0.0, 0.0, -1.0e-6, 25.0, 90.0, 30.0, 5.0, 1, 1, 0"
```

However, if the cumulative slip and dilation tendency (D_ST_C, D_DT_C) should be computed, n3=1. Then, the two zones that should be compared need to be included. Please note, that the zone names are case-sensitive. The command string syntax is then:

```
"SDT, xtr, xpl, ytr, ypl, ssf, fst, fdp, fr, co, n1, n2, n3, zn1, zn2"
```

exemplified by:

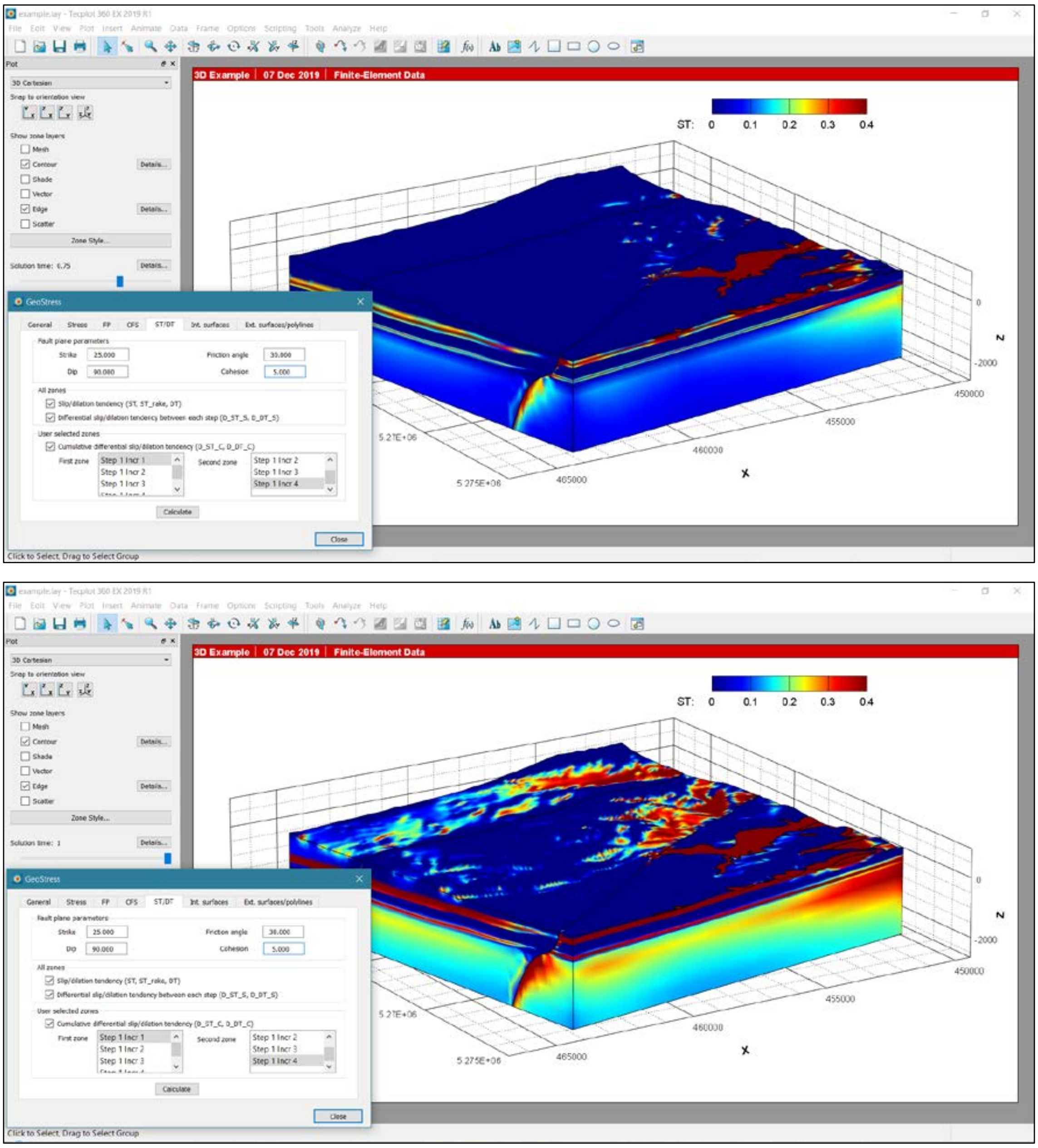
```
"SDT, 90.0, 0.0, 0.0, 0.0, -1.0e-6, 25.0, 90.0, 30.0, 5.0, 1, 1, 1, Step 1
Incr 1, Step 1 Incr 4"
```

4.5.3 Application examples

Figure 4.5-2 shows the *ST* values as a contour plot. Both *ST* values are calculated for vertical faults with a strike of 25°N almost optimally oriented in the given stress field. The upper figure is for *Solution Time* 0.75 (increment 3) and the lower one for *Solution Time* 1.0 (increment 4).

Note that the interpretation of the resulting colour distribution is potentially misleading. E.g. the higher or lower *ST* or *D_ST* values near the faults that are implemented in the model do not indicate that the fault has been brought relatively closer or further away from failure. The calculation of the *ST* values is not based on the geometry of the faults that are implemented in the model, but to generic faults that are almost perpendicular to them. Thus, the resolved shear and normal stresses are very different.

If the user changes the parameter of fault settings or the rock properties and re-calculate e.g. the *ST* values, the results in all frames are changing on the fly. This enables the user of *GeoStress* to perform a fast and efficient analysis of the model results.



☐ Mesh
☑ Contour
☐ Shade
☐ Vector
☑ Edge
☐ Scatter

Solution time: 0.75

Fault plane parameters
Strike 25.000
Dip 90.000
Friction angle 30.000
Cohesion 5.000

All zones
☑ Slip/dilation tendency (ST, ST_rake, DT)
☑ Differential slip/dilation tendency between each step (D_ST_S, D_DT_S)

User selected zones
☑ Cumulative differential slip/dilation tendency (D_ST_C, D_DT_C)
First zone: Step 1 Incr 1
Second zone: Step 1 Incr 4

☐ Mesh
☑ Contour
☐ Shade
☐ Vector
☑ Edge
☐ Scatter

Solution time: 1

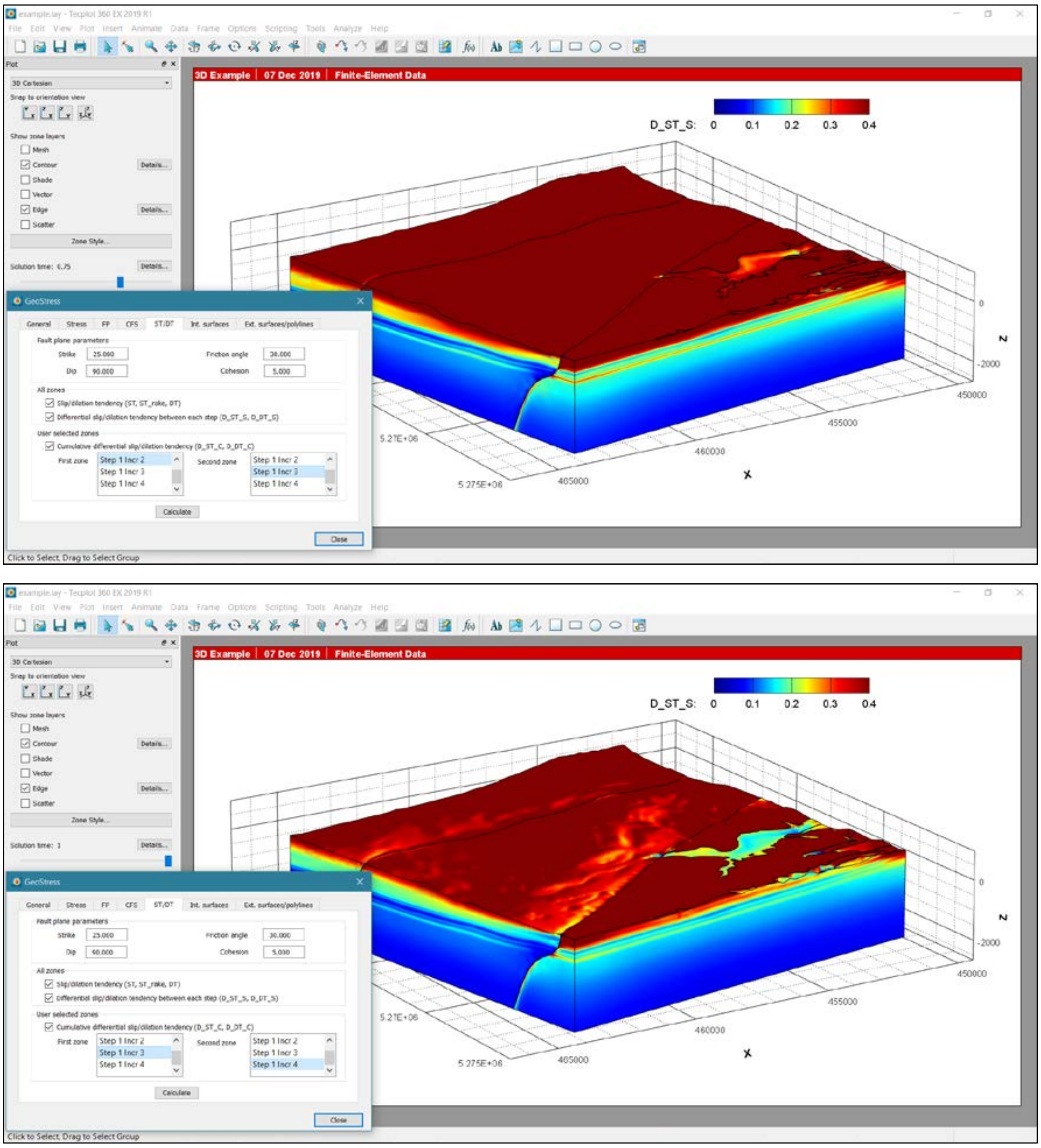
Fault plane parameters
Strike 25.000
Dip 90.000
Friction angle 30.000
Cohesion 5.000

All zones
☑ Slip/dilation tendency (ST, ST_rake, DT)
☑ Differential slip/dilation tendency between each step (D_ST_S, D_DT_S)

User selected zones
☑ Cumulative differential slip/dilation tendency (D_ST_C, D_DT_C)
First zone: Step 1 Incr 1
Second zone: Step 1 Incr 4

Fig. 4.5-2: Slip tendency calculation.

Both figures show the *ST* value calculated for vertical faults (dip=90°) that strike 25°N. The upper figure displays the ST value at *Solution Time* 0.75 (increment 3), the lower figure at *Solution Time* 1.0 (increment 4).



Upper screenshot:

Solution time: 0.75

Fault plane parameters
Strike 25.000
Dip 90.000
Friction angle 30.000
Cohesion 5.000

All zones
☑ Slip/dilation tendency (ST, ST_rake, DT)
☑ Differential slip/dilation tendency between each step (D_ST_S, D_DT_S)

User selected zones
☑ Cumulative differential slip/dilation tendency (D_ST_C, D_DT_C)
First zone: Step 1 Incr 2
Second zone: Step 1 Incr 3

Lower screenshot:

Solution time: 1

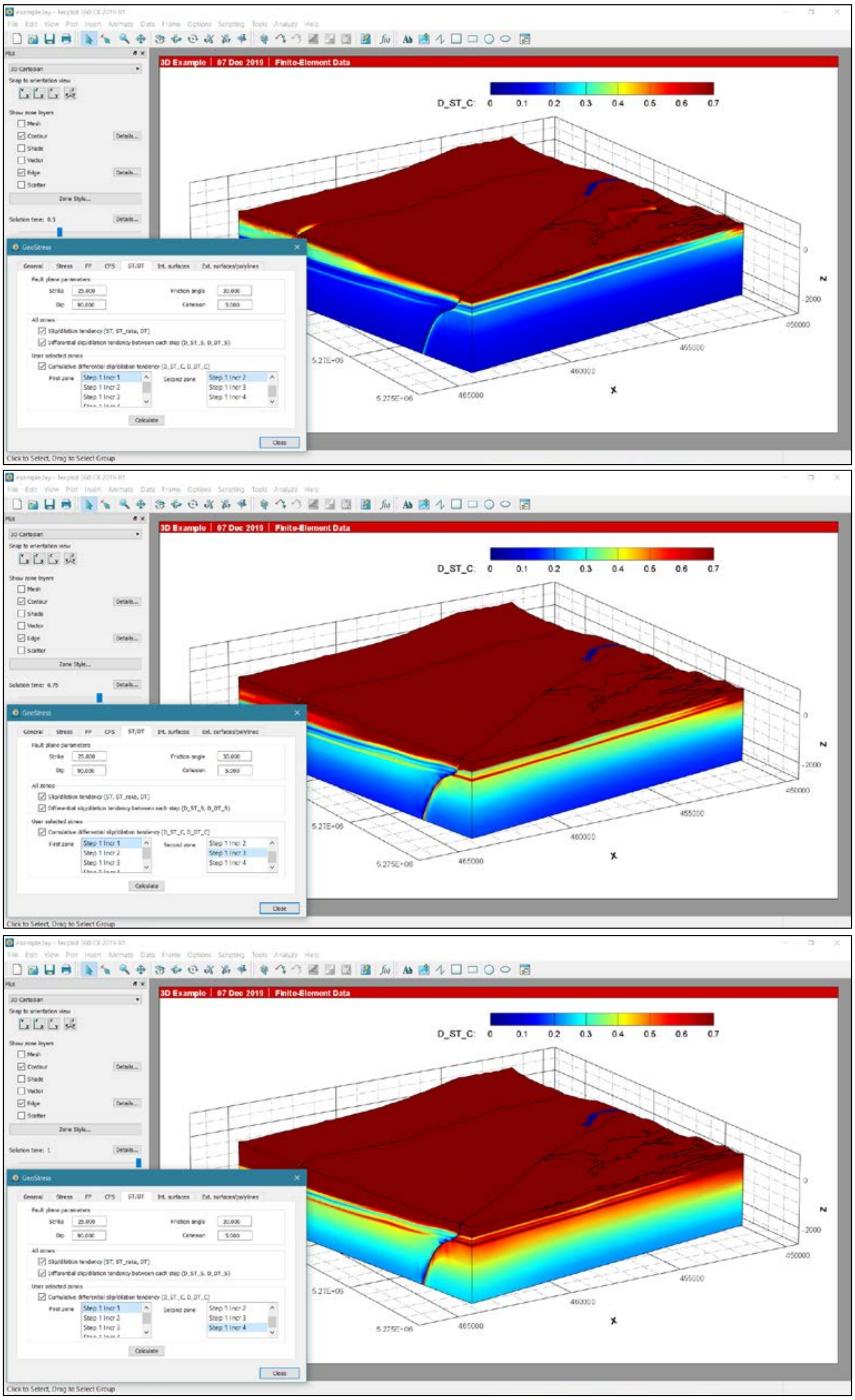
Fault plane parameters
Strike 25.000
Dip 90.000
Friction angle 30.000
Cohesion 5.000

All zones
☑ Slip/dilation tendency (ST, ST_rake, DT)
☑ Differential slip/dilation tendency between each step (D_ST_S, D_DT_S)

User selected zones
☑ Cumulative differential slip/dilation tendency (D_ST_C, D_DT_C)
First zone: Step 1 Incr 3
Second zone: Step 1 Incr 4

Fig. 4.5-3: Differential slip tendency between all *Zone*s.

The upper figure shows *D_DT_S* in MPa for Step 1 Incr3 - Step1 Incr2. The lower figure shows *D_ST_S* in MPa for Step 1 Incr4 - Step1 Incr3. The *D_ST_S* values are always the difference to the previous *Zone* and written into the second *Zone*.



Top: Solution time: 0.5

Fault plane parameters
Strike 25.000 Friction angle 30.000
Dip 90.000 Cohesion 5.000

All zones
☑ Slip/dilation tendency (ST, ST_ratio, DT)
☑ Differential slip/dilation tendency between each step (D_ST_S, D_DT_S)

User selected zones
☑ Cumulative differential slip/dilation tendency (D_ST_C, D_DT_C)
First zone: Step 1 Incr 1 Second zone: Step 1 Incr 2

Middle: Solution time: 0.75 — First zone: Step 1 Incr 1 Second zone: Step 1 Incr 3

Lower: Solution time: 1 — First zone: Step 1 Incr 1 Second zone: Step 1 Incr 4

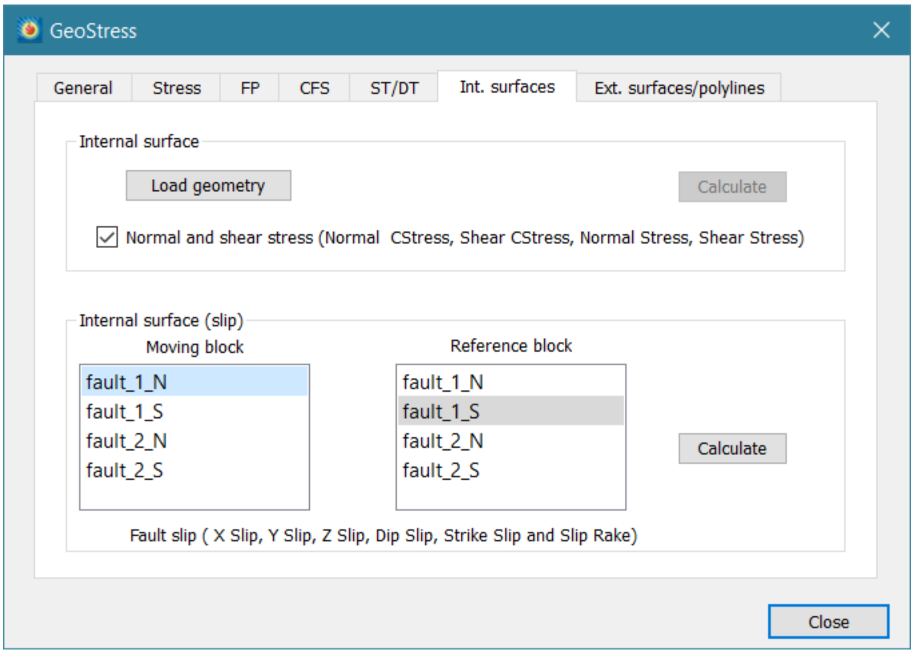
Fig. 4.5-4: Cumulative differential slip tendency between the user defined *Zones*.

D_ST_C in MPa for three steps. Top: *D_ ST _C* for Step1 Incr2 – Step 1 Incr1; Middle: Step1 Incr3 - Step 1 Incr2; Lower: Step1 Incr4 - Step 1 Incr3. *D_ ST _C* values presented in a *Zone* is always the difference to the selected first *Zone* and cumulative value.

4.6 Internal Surfaces

Generally, the Add-on *GeoStress* distinguishes between two surface types: Internal and External. Internal surfaces are the ones that are implemented in the geomechanical-numerical model where relative sliding is allowed to occur. In Abaqus these surfaces are so-called contact surfaces. The potential sliding along these surfaces changes the stress field at the interface and in its vicinity. In contrast to this, external surfaces are explicitly not part of the geomechanical-numerical model geometry. They are only used to visualize the model results and derived variables at desired surfaces, i.e. they are interpolated on these imported surfaces. This option is described in section 4.7.

With the page *Internal Surfaces* the user can display variables that exist in the data set onto the internal surface. The geometry of these surfaces has to be imported from a text file with a format that will be described below. This import generates a new set of *Zone*s for this geometry only and the results can be displayed over time (increments/steps) on these surfaces without showing the whole volume. For internal surfaces the user can also estimate the fault slip, i.e. the relative displacement of the blocks (Fig. 4.6-1).



☑ Normal and shear stress (Normal CStress, Shear CStress, Normal Stress, Shear Stress)

Fig. 4.6-1: Page *Internal* Surfaces.

The page is divided into two sections. In the upper section the geometry files (*.txt) of the internal surface has to be loaded and the shear and normal stresses are calculated. In the lower section the relative movement between the two blocks of an internal contact can be estimated.

4.6.1 Internal surfaces represented as pairs of contact surfaces

In Abaqus the relative movement of blocks along a fault is implemented with a pair of contact surfaces. These so-called internal surfaces are made of faces of 3D finite elements within the 3D model geometry. A contact surface is a non-linearity of the model where the stress field changes due to the displacement along this surface. However, Tecplot 360 EX does not provide sufficient information about the geometry of the internal surface and thus it has to be imported again. This input file that describes the faces and nodes on the internal surface must have the exact same node numbering and element definition as in the Abaqus model. Only element faces that are described with three or four nodes are accepted.

Example of the geometry input file for internal surfaces

```
** ABAQUS Input Deck Generated by HyperMesh Version: 12.0.0.85
** Generated using HyperMesh-Abaqus Template Version: 12.0
**
*NAME fault_1_S
*NODE
1,  678154.26845481,  275868.40204563,  -2500.000000000
2,  678177.54278354,  275736.40715276,  -1699.950575759
.
.
974791,  683685.31005814,  276833.62794856,  134.70187273715
978455,  684980.76375911,  277172.53241215,  84.299970524759
.
.
*ELEMENT,TYPE=S3
761261,    104225,    104224,    104226
761311,    169371,    169229,    169227
828005,    104225,    104224,    104226
.
.
*ELEMENT,TYPE=S4
762143,    191459,    168028,    168647,    186086
762204,    366574,    344480,    342914,    367161
.
.
**
```

The structure of the input file *.txt must contain the following three key words: *NAME, *NODE, *ELEMENT, TYPE=S4 (or S3).

- Letters and/or numbers that follow the key word *NAME   determine the name of the new *Zone* that is generated in Tecplot 360 EX, but only the first 25 characters are used.
- After the key word *NODE follows the original node numbering from the finite element mesh of the model and the coordinates of the nodes. The coordinates are in fact not used since Tecplot 360 EX knows from all original node numbering the coordinates. Field separators can be comma, tabs or blanks.
- After the key word *ELEMENT, TYPE=S3 (or S4) the (original) element-node assignments using the previously given nodes has to be provided.
- Comment line must begin with ** and the input file must end with a comment line.

Normal and shear stresses (variables *Normal Stress* and *Shear Stress*) can be estimated for the individual segments i.e. the element faces that define the contact surfaces. *GeoStress* calculates the normal vector of the element face from its nodes and estimates from the stress tensors at the nodes a mean stress tensor. In this context, the term element means only the appropriate face of a 3D element. Using the normal vector and the mean stress *GeoStress* resolves *Normal Stress* and *Shear Stress* and stores them as element face centred variables.

Abaqus also provides for contact surfaces the shear and normal stress in the *.odb output file. These variables are named *CPRESS* (Normal Stress), *CSHEAR1*, *CSHEAR2* and *CSHEAR3* (the three components of the shear vector given in a local coordinate system; *CSHEAR3* points into the normal vector of the element face and is zero). Tecplot 360 EX is loading these Abaqus nodal variables and renames them to *Contact Pressure* for the normal stress component, and

Contact, X Component Traction and Contact, Y Component Traction (the variable *Contact, Z Component Traction* is not loaded by Tecplot 360 EX). These two components describe the shear stress on the fault segment. The Add-on *GeoStress* also uses these values to calculate appropriate element face centred variables *Normal CStress* and *Shear CStress*.

The values from Abaqus and *GeoStress* can deviate from each other since Abaqus is probably using the stress tensor values from the integration points or the results from the equilibrium solution procedure on the contact surface. Thus, at surfaces with high curvature the deviation can be significant. However, in these cases the artefacts that are generated by the modelling itself are probably larger and more unrealistic than the deviation between the two approaches.

The bug in Tecplot 360 EX 2015 R2 where *Contact Pressure* was only loaded for the first contact surface, but not for other contact surfaces in the model has been fixed in Tecplot 360 EX 2019 R1 and now values are provided for all contact surfaces. In general, the determined values from *GeoStress* should be slightly smoother than the values from Abaqus, but the deviations are clearly within the high numerical uncertainties of the implementation of faults as contact surfaces.

GeoStressCmd: In *GeoStressCmd*, general information on the stress scaling factor and the plot scaling factor are required. This is followed by the assertion that normal and shear stress as well as the normal and shear contact stress are computed. Then, the full path to the file that contains the surface is set. The command string syntax is:

```
"IntSurface, ssf, psf, n1, fn"
```

exemplified by:

```
"IntSurface, -1.0e-6, 1.0, 1, C:\\Desktop\\Tecplot_Example\\fault_1_N.txt".
```

4.6.2 Fault slip

In the list boxes *Moving block* and *Reference block* the imported two surfaces of a contact pair can be used to calculate their relative displacements. One side is assigned to the *Moving block* and the other to the *Reference block*. Note that it is compulsory for the calculation of the fault kinematic that the surface normal (the normal for each element face that defines the surface) is facing outward from the contact surface pair. The orientation of the normal is determined through the order of element-node assignment according to the right-hand rule in the input file. In chapter 4 we present a way to check the orientation of the normal vectors in Tecplot 360 EX and in HyperMesh. *GeoStress* is calculating the following kinematic variables:

- *X Slip*, *Y Slip* and *Z Slip*: Vector (in the {*X, Y, Z*}-System) of the relative displacement of the moving block with respect to the reference block. This is needed from Tecplot 360 EX in order to display the slip vector in the Tecplot system.
- *Dip Slip*: Downwards oriented slip component on the surface.
- *Strike Slip*: Is given on the fault in direction of the strike slip component.
- *Slip Rake*: Slip direction on the surface (see failure plane variable for the definition of strike and rake).

These variables are stored in the *Zone*s of the *Reference block*. Each increment is presented in one *Zone* and thus, if an increment is given for a special model time, fault slip rates can be derived. In chapter 4 we show how a fault slip variable can be determined in Tecplot 360 EX using the option *Data/Alter/Specify Equations...*

GeoStressCmd: In GeoStressCmd, general information on the stress scaling factor and the plot scaling factor are required. This is followed by the surface name of the moving block (sn1) and the reference block (sn2). The command string syntax is:

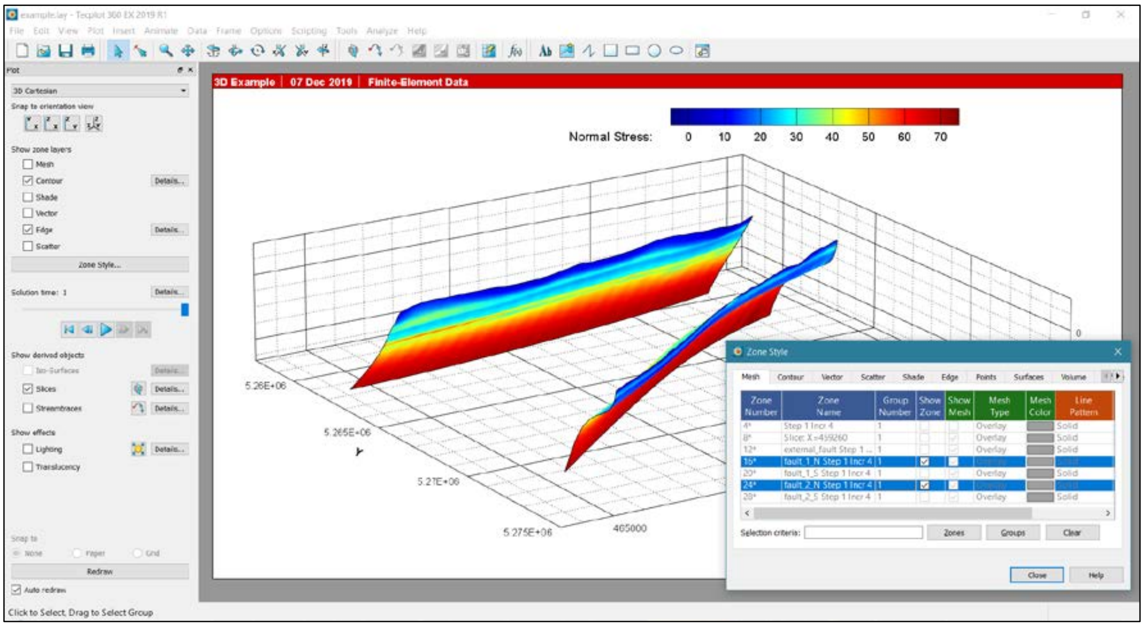
```
"IntSurfaceSlip, ssf, psf, sn1, sn2"
```

exemplified by:

```
"IntSurfaceSlip, -1.0e-6, 1.0, fault_1_N, fault_1_S"
```

4.6.3 Application examples for internal surfaces

In order to generate the results displayed in Figure 4.6-2 (*Normal Stress*) and Figure 4.6-3 (*Dip Slip*) the geometry of the two surfaces that define the contact pair in the example model are provided in the geometry folder with the names fault_1_S.txt, fault_1_N.txt, fault_2_S.txt and fault_2_N.txt. Fig. 4.6-2 shows that the normal stress is increasing with depth, but it also shows that this increase is not linear due to the curvature of the fault and due to the displacement boundary condition that is producing different stresses on the individual segments of the fault. Fig. 4.6-3 shows that the dip slip also is not equally distributed over the surface of the fault. The dip slip is displayed on the southern surface. I.e., the values displayed show the displacement in dip slip direction with respect to the fixed northern block.



Plot

3D Cartesian

Snap to orientation view

Show zone layers

☐ Mesh

☑ Contour

☐ Shade

☐ Vector

☑ Edge

☐ Scatter

Zone Style...

Solution time: 1

Show derived objects

☐ Iso-Surfaces

☑ Slices

☐ Streamtraces

Show effects

☐ Lighting

☐ Translucency

Snap to

● None ○ Paper ○ Grid

Redraw

☑ Auto redraw

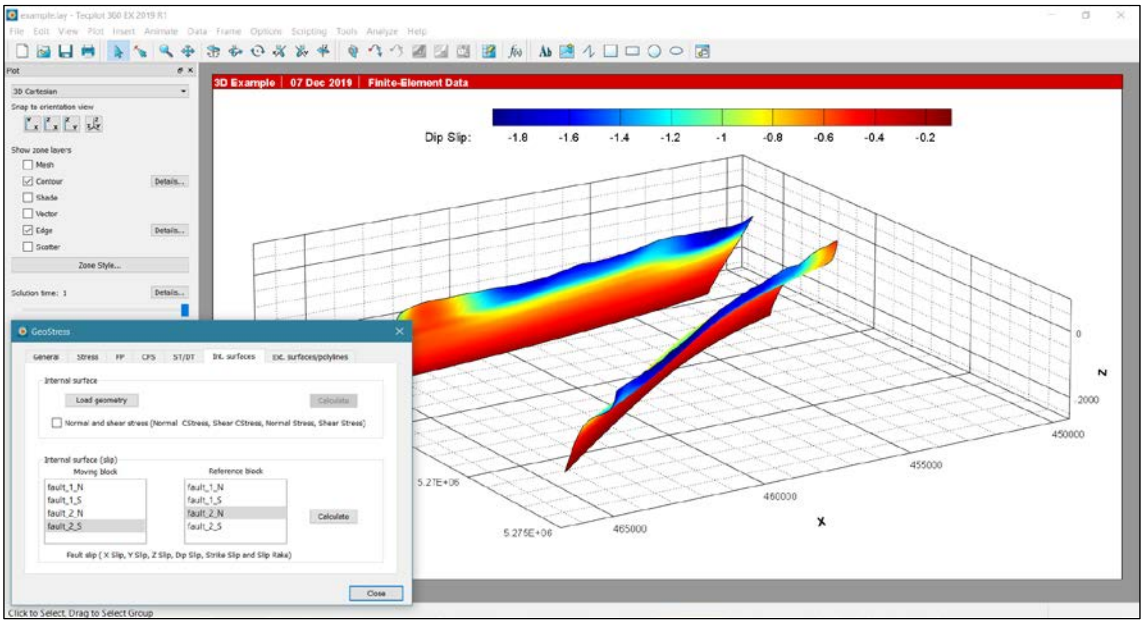
Click to Select, Drag to Select Group

Zone Number	Zone Name	Group Number	Show Zone	Show Mesh	Mesh Type	Mesh Color	Line Pattern
4*	Step 1 Incr 4	1	☐	☐	Overlay	▨	Solid
8*	Slice: X=459260	1	☐	☑	Overlay	▨	Solid
12*	external_fault Step 1 ...	1	☐	☑	Overlay	▨	Solid
16*	fault_1_N Step 1 Incr 4	1	☑	☐	Overlay	▨	Solid
20*	fault_1_S Step 1 Incr 4	1	☐	☑	Overlay	▨	Solid
24*	fault_2_N Step 1 Incr 4	1	☑	☐	Overlay	▨	Solid
28*	fault_2_S Step 1 Incr 4	1	☐	☑	Overlay	▨	Solid

Selection criteria:

Fig. 4.6-2: Normal stress distribution on an internal surface.

The *Normal stress* is calculated for each increment on the surface. Note that the *Normal Stress* does not exist in the model volume, but only for the *Zones* where the internal and external surfaces are stored.



☐ Mesh
☑ Contour
☐ Shade
☐ Vector
☑ Edge
☐ Scatter

Solution time: 1

Internal surface

☐ Normal and shear stress (Normal CStress, Shear CStress, Normal Stress, Shear Stress)

Internal surface (slip)

Moving block	Reference block
fault_1_N	fault_1_N
fault_1_S	fault_1_S
fault_2_N	fault_2_N
fault_2_S	fault_2_S

Fault slip (X Slip, Y Slip, Z Slip, Dip Slip, Strike Slip and Slip Rake)

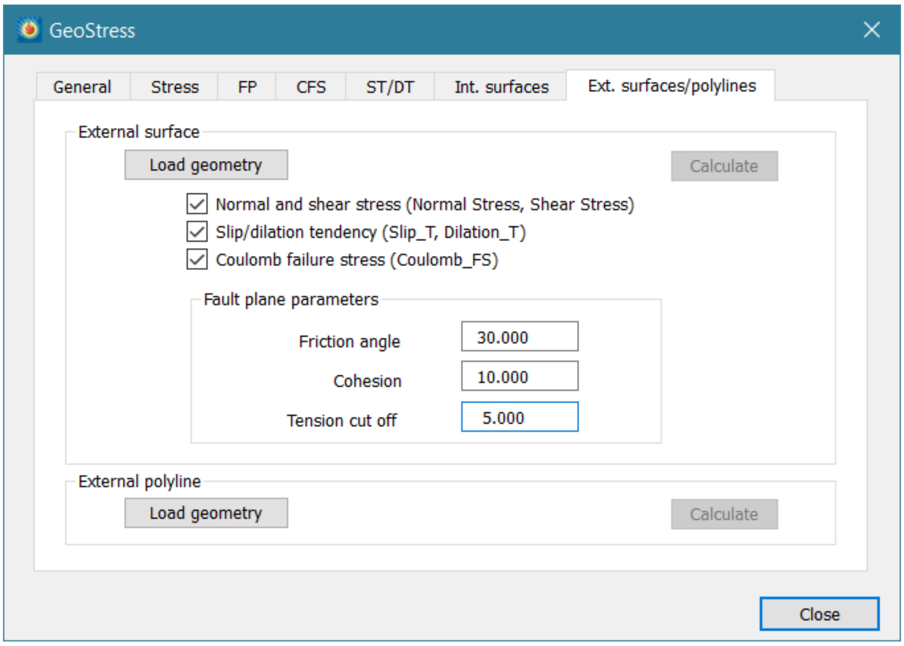
Fig. 4.6-3: Dip slip distribution on an internal surface.

The *Dip slip* is calculated for each increment on the surface. Note that the *Dip Slip* does not exist in the model volume, but only for the *Zone*s where internal surfaces are stored. External surfaces cannot have a relative displacement as they are not part of the model.

4.7 External surfaces and polylines

With the page *External Surfaces/Polylines* the user can map all variables that exist at the nodes of the finite element mesh including the variables that have been generated with *GeoStress* onto external surfaces or polylines. Note that an external surface has no impact on the model results since it was not present as a contact surface during computation. In that sense, it is a generic potential fault on which the model results are interpolated. The same holds on for a polyline that represents e.g. a planned borehole drilling path way.

The page is divided into two sections (Fig. 4.7-1). In the upper section an external surface can be loaded and in the lower section a polyline. For external surfaces (in addition to the automatic calculation of stress variables from the model volume) the user can choose to calculate the *Normal Stress* and *Shear Stress* as well as the values for Slip and dilation tendency (*Slip_T, Dilation_T*) and the Coulomb failure stress (*Coulomb_FS*) using the fault plane parameter and the maximum shear stress values according to the loaded fault geometry. This allows the computation of Slip tendency or the Coulomb Failure Stress, which are highly dependent on the fault orientation, for highly complex fault geometries. Thereby patches of increased criticality due to the fault orientation can be identified. For both geometries, new *Zone*s are generated and can be displayed separately.



☑ Normal and shear stress (Normal Stress, Shear Stress)
☑ Slip/dilation tendency (Slip_T, Dilation_T)
☑ Coulomb failure stress (Coulomb_FS)

Friction angle 30.000
Cohesion 10.000
Tension cut off 5.000

Fig. 4.7-1: Page External *Surfaces/Polylines*.

The page is divided in two sections. In the upper section, external surfaces can be loaded to calculate all values that also exist in the volume for these and to calculate fault normal and shear stresses, slip and dilation tendency (*Slip_T, Dilation_T*) as well as Coulomb failure stress (*Coulomb_FS*). In the lower section an external polyline (e.g. representing a borehole pathway) can be loaded. For both geometries, new *Zone*s are generated.

4.7.1 External surfaces for result visualisation

External surfaces are geometries that are not a part of the model. Thus, they are only used in order to display a posteriori variables from the model volume onto these surfaces. The format of the surface description is the same as for the internal surface except that only finite element meshes with triangles are allowed.

Example of the geometry input file for external surfaces

```
** ABAQUS Input Deck Generated by HyperMesh Version: 12.0.0.85
** Generated using HyperMesh-Abaqus Template Version: 12.0
**
*NAME External Fault
*NODE
982773,  688871.28373095,  276072.97816205,  -251.6813894955
982776,  688640.90023015,  276266.34877857,  -271.4864931561
.
.
.
982777,  688562.55565372,  276328.72343553,  -278.5207494841
985821,  682612.87832328,  277777.36569695,  -893.3797783981
*ELEMENT,TYPE=S3
761141,    985636,    985821,    985670
761139,    985509,    985820,    985504
.
.
.
755302,    982878,    982879,    982880
755300,    982855,    982856,    982857
**
```

Three key words determine the structure of the text file *.txt: *NAME, *NODE, *ELEMENT, TYPE=S3.

- Letters and/or numbers that follow the key word *NAME determine the name of the new *Zone* that is generated in Tecplot 360 EX, but only the first 25 characters are used.
- After the key word *NODE follows directly the node numbering of the model mesh and the coordinates of the nodes. Field separators can be commas, tabs or blanks.
- After the key word *ELEMENT, TYPE=S3 the element-node assignments, i.e. the definition of the element of the previously given nodes has to be provided.
- Comment line must begin with ** and the input file must end with comment line.

The variables for the normal stress and the shear stress (*Normal Stress*, *Shear Stress*) are calculated using the normal vector of the 2D surface element and the mean of the stress tensor at the face nodes. They are determined from the original Abaqus stress components *SXX*, *SYY*, *SZZ*, *SXY*, *SYZ* and *SZX* by the Tecplot 360 EX interpolation procedure. Note that the output variable *Normal Stress* and *Shear Stress* are 2D element centred variables.

GeoStressCmd: In *GeoStressCmd*, general information on the stress scaling factor and the plot scaling factor are required. This is followed by definition of the variables that should be derived. Normal and shear stress on the plane (n1), Slip and Dilation tendency (n2) and Coulomb Failure Stress (n3) can be selected. If slip and dilation tendency are selected, information on the friction angle (fr) and cohesion (co) are required. If Coulomb Failure Stress is selected, information on the friction angle (fr), cohesion (co) and tensile strength of the rock (te) are required. This information is followed by the full path to the file that contains the surface. The command string syntax is:

```
"External Surface, ssf, psf, n1, n2, n3 [, fr, co [, te]], fn"
```

exemplified by:

```
"ExtSurface, -1.0e-6, 1.0, 1, 1, 0, 30.0, 10.0,
C:\\Desktop\\Tecplot_Example\\external_surface.txt"
```

4.7.2 External polyline for result visualisation

External polylines are also geometries that were not part of the model. Thus, they are only used to display a posteriori variables from the model volume onto these lines. The format of the polyline description is described in the following.

Example of the geometry input file for external polylines

```
*NAME Polyline_12345
** any comment
*NODE
1, 458195 ,5266455,  0
2, 458198 ,5266457,  -20
.
.
.
900, 460360 ,5267705, -2495
901, 460360 ,5267705, -2500
**
```

Two key words determine the structure of the text file *.txt: *NAME and *NODE.

- Letters and/or numbers that follow the key word *NAME determine the name of the new *Zone* that is generated in Tecplot 360 EX, but only the first 25 characters are used.
- After the key word *NODE follow in each line the node numbering and the coordinates X, Y, Z of the nodes. Field separators can be tabs, commas or blanks.
- Comment lines must begin with ** and the input file must end with a comment line.

All nodal variables that exist in the model are interpolated onto the polyline.

GeoStressCmd: In *GeoStressCmd*, general information on the stress scaling factor and the plot scaling factor are required. This is followed by the full path to the file that contains the polyline. The command string syntax is:

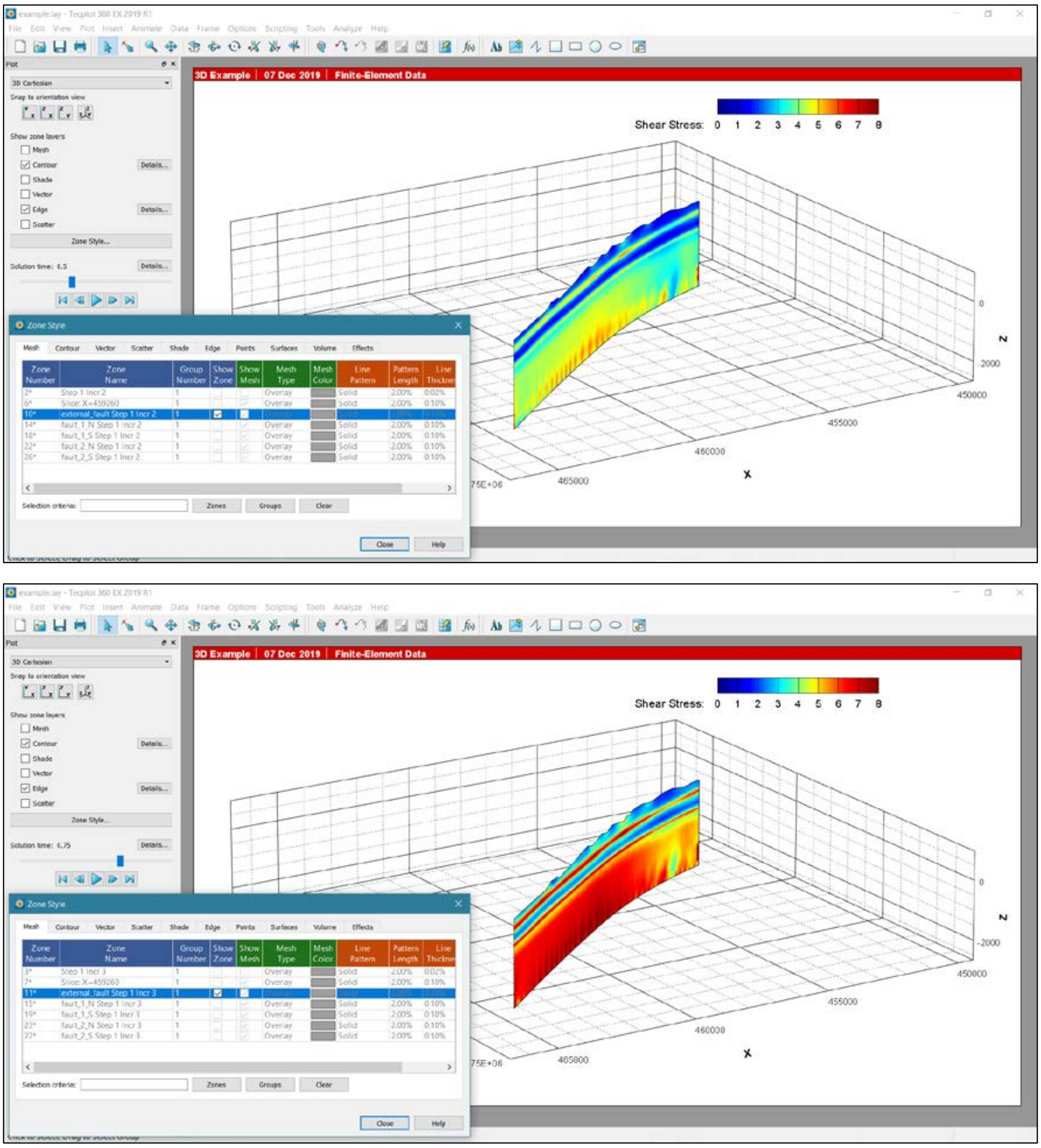
```
"ExtPolyline, ssf, psf, fn"
```

exemplified by:

```
"ExtPolyline, -1.0e-6, 1.0,
C:\\Desktop\\Tecplot_Example\\external_polyline.txt"
```

4.7.3 Application example for external surfaces

To generate the results displayed in Figure 4.7-2 the geometry of an external surface is provided in the geometry folder with the name external_surface.txt. For the visualization of the shear stress the newly created *Zone* is activated in the *Zone Style* dialog box and the other *Zone*s must be deactivated.



Solution time: 0.5

Zone Number	Zone Name	Group Number	Show Zone	Show Mesh	Mesh Type	Mesh Color	Line Pattern	Pattern Length	Line Thickne
2*	Step 1 Incr 2	1	☐	☐	Overlay	■	Solid	2.00%	0.02%
6*	Slice: X=459260	1	☐	☑	Overlay	■	Solid	2.00%	0.10%
10*	external_fault Step 1 Incr 2	1	☑	☑	[illegible]	■	[illegible]	[illegible]	[illegible]
14*	fault_1_N Step 1 Incr 2	1	☐	☑	Overlay	■	Solid	2.00%	0.10%
18*	fault_1_S Step 1 Incr 2	1	☐	☑	Overlay	■	Solid	2.00%	0.10%
22*	fault_2_N Step 1 Incr 2	1	☐	☑	Overlay	■	Solid	2.00%	0.10%
26*	fault_2_S Step 1 Incr 2	1	☐	☑	Overlay	■	Solid	2.00%	0.10%

Solution time: 0.75

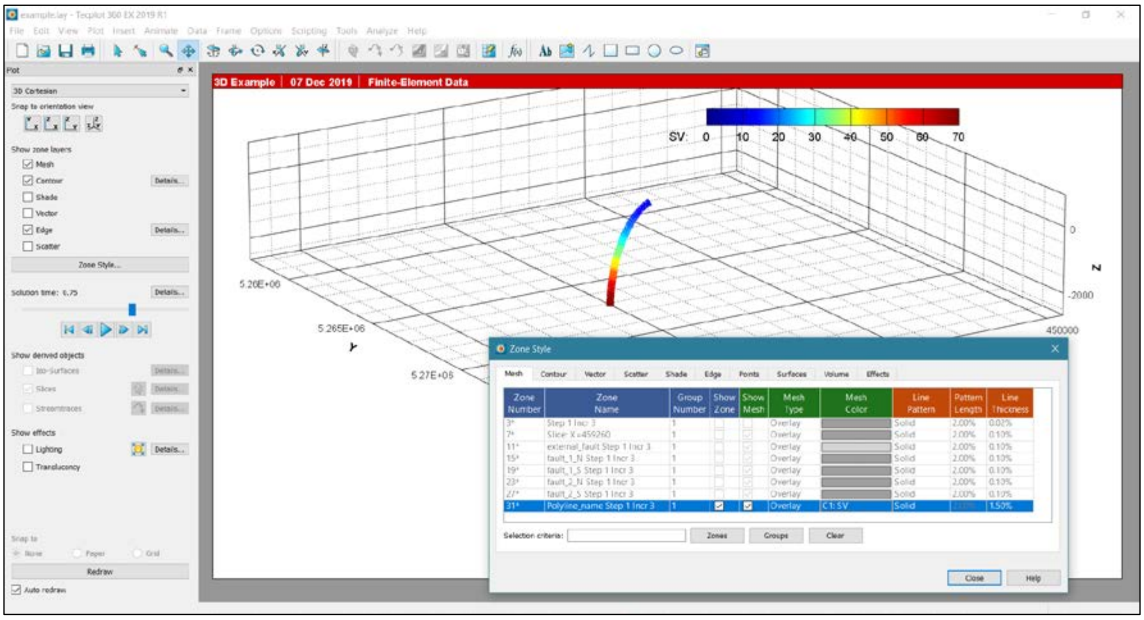
Zone Number	Zone Name	Group Number	Show Zone	Show Mesh	Mesh Type	Mesh Color	Line Pattern	Pattern Length	Line Thickne
3*	Step 1 Incr 3	1	☐	☐	Overlay	■	Solid	2.00%	0.02%
7*	Slice: X=459260	1	☐	☑	Overlay	■	Solid	2.00%	0.10%
11*	external_fault Step 1 Incr 3	1	☑	☑	[illegible]	■	[illegible]	[illegible]	[illegible]
15*	fault_1_N Step 1 Incr 3	1	☐	☑	Overlay	■	Solid	2.00%	0.10%
19*	fault_1_S Step 1 Incr 3	1	☐	☑	Overlay	■	Solid	2.00%	0.10%
23*	fault_2_N Step 1 Incr 3	1	☐	☑	Overlay	■	Solid	2.00%	0.10%
27*	fault_2_S Step 1 Incr 3	1	☐	☑	Overlay	■	Solid	2.00%	0.10%

Fig. 4.7-2: Shear stress distribution on an external surface.

The *Shear Stress* is calculated for each increment on the surface. The upper figure is for *Solution Time* 0.5 (Step1 Inc2), the lower figure shows the results for *Solution Time* 0.75 (Step1 Inc3). Note the *Shear Stress* does not exist in the model volume, but only for the *Zone*s where internal and external surfaces are stored.

4.7.4 Application example for external polylines

To generate the results displayed in Figure 4.7-2 the geometry of an external polyline is provided in the geometry folder with the name external_polyline.txt. For the visualization of the vertical stress S_V as an example the newly created *Zone* is activated in the *Zone Style* dialog box. To display the values the mesh colour of the polyline must be changed from a single colour to Multi 1 colour setting (see Fig. 4.7-2). Furthermore, the line thickness of the mesh should be increased to 1.5% and the Edge display option should be turned off.



Zone Style

Mesh | Contour | Vector | Scatter | Shade | Edge | Points | Surfaces | Volume | Effects

Zone Number	Zone Name	Group Number	Show Zone	Show Mesh	Mesh Type	Mesh Color	Line Pattern	Pattern Length	Line Thickness
3*	Step 1 Incr 3	1	☐	☐	Overlay	▇	Solid	2.00%	0.02%
7*	Slice: X=459260	1	☐	☑	Overlay	▇	Solid	2.00%	0.10%
11*	external_fault Step 1 Incr 3	1	☐	☑	Overlay	▇	Solid	2.00%	0.10%
15*	fault_1_N Step 1 Incr 3	1	☐	☑	Overlay	▇	Solid	2.00%	0.10%
19*	fault_1_S Step 1 Incr 3	1	☐	☑	Overlay	▇	Solid	2.00%	0.10%
23*	fault_2_N Step 1 Incr 3	1	☐	☑	Overlay	▇	Solid	2.00%	0.10%
27*	fault_2_S Step 1 Incr 3	1	☐	☑	Overlay	▇	Solid	2.00%	0.10%
31*	Polyline_name Step 1 Incr 3	1	☑	☑	Overlay	C1: SV	Solid	[illegible]	1.50%

Selection criteria:

Fig. 4.7-3: Vertical stress along an external polyline.

The vertical stress S_V is calculated for each increment along the polyline. The figure shows the SV for the fourth *Solution Time* 1.0 (Step1 Inc4).

Acknowledgements

The authors would like to thank the NAGRA (National Cooperative for the Disposal of Radioactive Waste in Switzerland) for their support to develop this tool. We also received co-funding from Germany's Federal Ministry for Education and Research under the FONA/GEO:N programme under Grant agreement No. 03G0869A (Project SUBI) and from the "Integrity of nuclear waste repository systems—Cross-scale system understanding and analysis (iCross)" project by the Federal Ministry of Education and Research (project number 02NUK053D), the Helmholtz Association (Grant number SO-093), and the Helmholtz Centre Potsdam—GFZ German Research Center for Geosciences.

Furthermore, we thank Karsten Reiter and Tobias Hergert for testing *GeoStress* and their review of the manual and Kirsten Elger for final checking text layout.

References

Aki, K. and Richards, P.G. (1980). Quantitative Seismology, Vol. 1, W.H. Freeman and Co Ltd. https://doi.org/10.1002/gj.3350160110

Anderson, E.M. (1905). The dynamics of faulting, Trans. Edin. Geol. Soc. 8, 387-402. https://doi.org/10.1144/transed.8.3.387

Anderson, E.M. (1951). The dynamics of faulting and dyke formation with application to Britian, Oliver and Boyd.

Brady, B.H.G. and Brown, E.T. (2004), Rock Mechanics For Underground Mining, 3rd edtion, Kluwer Academic Publishers. https://doi.org/10.1007/978-1-4020-2116-9

Célérier, B. (1995). Tectonic regime and slip orientation of reactivated faults, Geophys. J. Int. 121, 143-161, https://doi.org/10.1111/j.1365-246X.1995.tb03517.x

Connolly, P. and Cosgrove, J. (1999). Prediction of static and dynamic fluid pathways within and around dilational jogs, In: Fractures, Fluid Flow and Mineralization, Vol. 155, Eds. K.J.W. McCaffrey, L. Lonergan and J.J. Wilinkson, 105-121, Geological Society, https://doi.org/10.1144/GSL.SP.1999.155.01.09

Engelder, T. (1992). Stress Regimes in the Lithosphere, p. 457, Princeton, NJ.

Engelder, T. (1994). Deviatoric Stressitis: A Virus Infecting the Earth Science Community, EOS Trans. 75, 209, 211-212, https://doi.org/10.1029/94EO00885

Friedmann, M. (1972). Residual elastic strain in rocks, Tectonophysics 15, 297-330, https://doi.org/10.1016/0040-1951(72)90093-5

Heidbach, O.; Tingay, M.; Barth, A.; Reinecker, J.; Kurfeß, D. and Müller, B. (2010). Global crustal stress pattern based on the World Stress Map database release 2008, Tectonophysics 482, 3-15, https://doi.org/10.1016/j.tecto.2009.07.023

Heidbach, O.; Hergert, T.; Reiter, K. and Giger, S. (2013). Local stress field sensitivity analysis - case study Nördlich Lägern, NAGRA Arb. Ber. NAB 13-88, pp. 50. URL: https://www.nagra.ch/de/cat/publikationen/arbeitsberichte-nabs/nabs-2013-81/downloadcenter.htm

Hergert, T.; Heidbach, O.; Reiter, K.; Giger, S. and Marschall, P. (2015). Stress field sensitivity analysis in a sedimentary sequence of the Alpine foreland, northern Switzerland, Solid Earth 6, 533-552, https://doi.org/10.5194/se-6-533-2015

Jaeger, J.C.; Cook, N.G.W. and Zimmermann, R.W. (2007) Fundamentals of Rock Mechanics, Blackwell Publishing. ISBN 978-0-632-05759-7

McGarr, A. and Gay, N.C. (1978). State of Stress in the Earth's Crust, Ann. Rev. Earth Planet. Sci. 6, 405-436, https://doi.org/10.1146/annurev.ea.06.050178.002201

Morris, A., D. A. Ferril, and D. B. Henderson (1996), Slip-tendency analysis and fault reactivation, Geology, 24(3), 275-278, https://doi.org/10.1130/0091-7613(1996)024<0275:STAAFR>2.3.CO;2

Neves, M.C.; Paiva, L.T. and Luis, J. (2009). Software for slip-tendency analysis in 3D: A plug-in for Coulomb, Comp. & Geosciences 35, 2345-2352, https://doi.org/10.1016/j.cageo-2009.03.008

Reiter, K. and Heidbach, O. (2014). 3-D geomechanical-numerical model of the contemporary crustal stress state in the Alberta Basin (Canada), Solid Earth, https://doi.org/10.5194/se-5-1123-2014

Schmitt, D.R.; Currie, C.A. and Zhang, L. (2012). Crustal stress determination from boreholes and rock cores: Fundamental principles, Tectonophys. 580, https://doi.org/10.1016/j.tecto.2012.08.029

Simpson, R.W. (1997). Quantifying Anderson's fault types, J. Geophys. Res. 102, 17909-17919, https://doi.org/10.1029/97JB01274

Stromeyer, D. and Heidbach, O. (2017): Tecplot 360 Add-on GeoStress, GFZ Data Services. https://doi.org/10.5880/wsm.2017.001

Ziegler, M., O. Heidbach, J. Reinecker, A. M. Przybycin, and M. Scheck-Wenderoth (2016). A multi-stage 3-D stress field modelling approach exemplified in the Bavarian Molasse Basin, Solid Earth, 7, 1365-1382, http:/doi.org/10.5194/se-7-1365-2016

Zoback, M. (2010). Reservoir Geomechanics, 2nd edition, Cambridge.