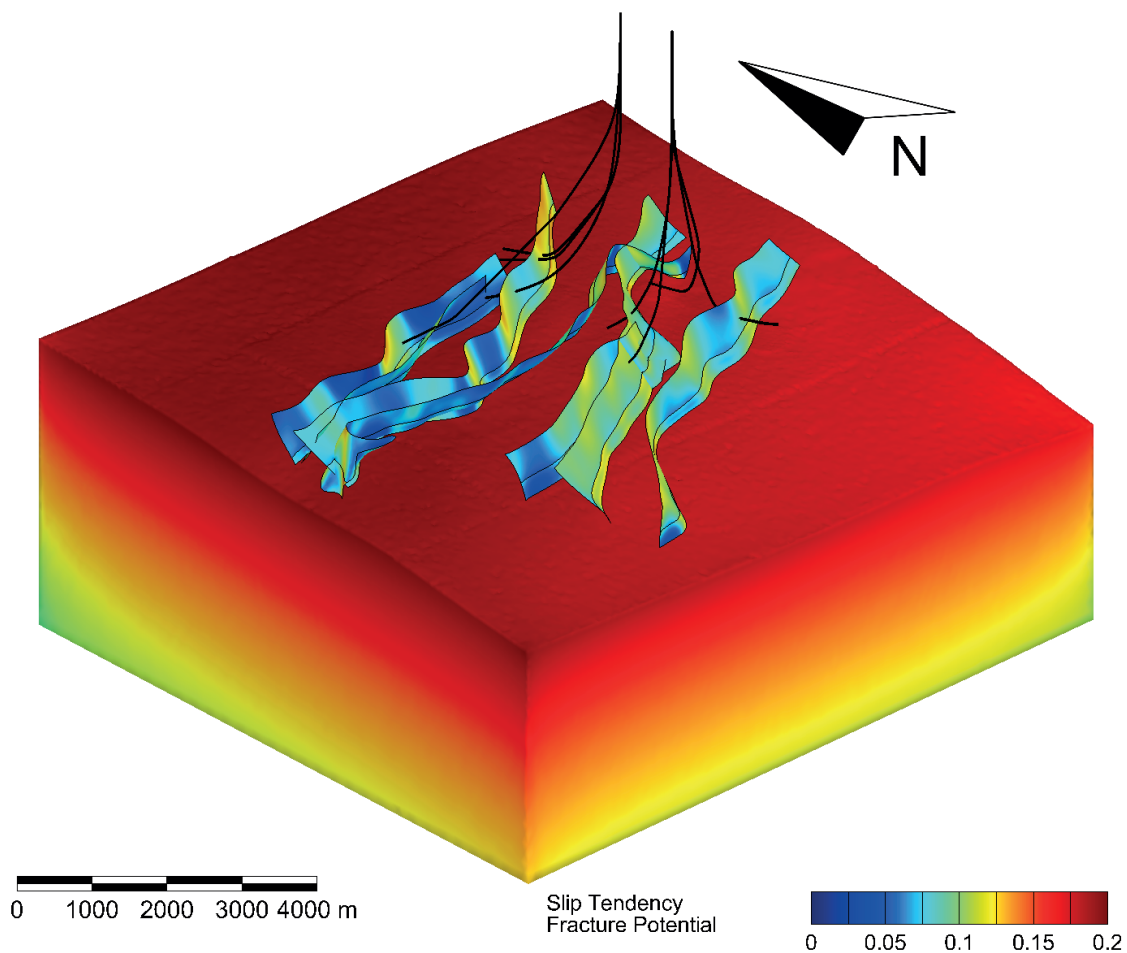


Manual of the tecplot® 360 ex Add-on *GeoStress*

Dietrich Stromeyer and Oliver Heidbach

March 2016 - Version 1.0



Summary

For the visualization and analysis of the stress field from 4D 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 stress tensor scalar and vector values such as the orientation and magnitude of the maximum and minimum horizontal stress, stress ratios, 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 has to 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 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*). Furthermore, *GeoStress* can estimate 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.

This report describes the technical details of the visualization tool, its usage and exemplifies its application using the results of a geomechanical-numerical model of the potential siting area Nördlich Lägern. The numerical solution is achieved with the finite element software Abaqus version 6.11. It also presents a number of special features of Tecplot 360 EX in combination with *GeoStress* that allow a professional and efficient analysis.

Zusammenfassung

Für die Visualisierung und Analyse des Spannungsfeldes aus thermo-hydro-mechanischen (THM) numerischen 4D Modellen sind zwei wesentliche technische Schritte notwendig. Zuerst müssen aus den sechs unabhängigen Komponenten des Spannungstensors skalare und vektorielle Größen wie z.B. die Orientierung und Magnitude der maximalen und minimalen horizontalen Spannung, Spannungsverhältnisse, differentielle Spannung oder Normal- und Scherspannung bezüglich einer beliebig vorgegebenen Fläche abgeleitet werden. Zweitens muss eine Geometrie angegeben werden, wie Querschnitte, Profile eines geplanten Bohrpfad oder eine beliebig vorgegebene Fläche, auf denen die Modellergebnisse interpoliert werden. Hierzu gehören auch die drei Feldgrößen Temperatur, Porendruck und der Verschiebungsvektor.

Zur Erleichterung und Automatisierung dieser Schritte wurde das Add-on *GeoStress* für die professionelle Visualisierungssoftware Tecplot 360 EX programmiert. Neben den vom Spannungstensor abgeleiteten Werten berechnet *GeoStress* auch die Coulombsche Versagensspannung (*Coulomb Failure Stress CFS*), das Bruchpotential (*Fracture Potential FP*) sowie die Tendenz zum Scherversagen und Öffnen (*Slip* und *Dilatation Tendency ST und DT*). Darüber hinaus bietet *GeoStress* die Möglichkeit, kinematische Variablen auf Störungen, die in dem Modell als Kontaktflächen implementiert sind, zu bestimmen (z.B. *dip slip* oder *rake vector*). Ebenso bietet das Tool die Möglichkeit die Tiefe unter Geländeoberkante exakt zu bestimmen.

In diesem Bericht werden die technischen Details des Add-on *GeoStress* vorgestellt und Beispiele für die Anwendung unter Verwendung der Ergebnisse des geomechanisch-numerischen Modells Nördlich Lägern beschrieben. Die numerische Lösung wurde mit der Finiten Elemente Software Abaqus v6.11 erreicht. Darüber hinaus werden spezielle, für geowissenschaftliche Modelle notwendige Nutzungsmöglichkeiten von Tecplot 360 EX in Kombination mit dem Add-On *GeoStress* vorgestellt, die eine professionelle und effiziente Analyse erlauben.

Table of Contents

Summary	I
Zusammenfassung.....	II
Table of Contents	III
List of Tables	IV
List of Figures.....	IV
1 Introduction	6
1.1 Motivation and objectives.....	6
1.2 Stress term definitions	7
2 Data visualisation with Tecplot 360 EX.....	9
2.1 Introduction	9
2.2 Basic concept and terminology of Tecplot 360 EX.....	10
2.3 The Add-on <i>GeoStress</i>	13
2.4 Saving Tecplot 360 EX format and output formats	13
2.4.1 Tecplot file formats *.lay and *.plt.....	13
2.4.2 Figure formats.....	14
2.4.3 Movie formats.....	15
2.4.4 ASCII data output format	16
2.5 Installation of the Tecplot 360 EX add-on <i>GeoStress</i>	17
2.6 Loading of the Abaqus odb- and inp-files	17
3 User interface of the add-on <i>GeoStress</i>	19
3.1 General settings	19
3.1.1 Orientation in North, East and Vertical down	19
3.1.2 Geo-coordinates.....	20
3.1.3 Stress scaling factor	20
3.1.4 Plot scaling factor	20
3.1.5 True vertical depth (TVD).....	21
3.1.6 Application examples	22
3.2 Stress value calculations	22
3.2.1 Theory and background information	22
3.2.2 Application examples	25
3.3 Fracture potential.....	26
3.3.1 Theory and background information	26
3.3.2 Application examples	28
3.4 Coulomb failure stress	30
3.4.1 Theory and background information	30
3.4.2 Application examples	31
3.5 Slip and dilation tendency	35

3.5.1	Theory and background information	35
3.5.2	Application examples	36
3.6	Internal Surfaces	40
3.6.1	Internal surfaces represented as pairs of contact surfaces	40
3.6.2	Application examples for internal surfaces	42
3.7	External surfaces and polylines	44
3.7.1	External surfaces for result visualisation	45
3.7.2	External polyline for result visualisation	46
3.7.3	Application example for external surfaces and polylines	46
3.7.4	Application example for external polylines	48
4	General tips for Tecplot 360 EX usage	49
4.1	Using Tecplot 360 EX layouts, frame styles and macros	49
4.2	Write ASCII data from a surface	51
4.3	Calculation of new variables with equations	51
4.4	Calculation of the normal vector of surfaces	52
5	References.....	55

List of Tables

Tab. 1.3-1:	Definition of stress terms.....	7
Tab. 3.2-1:	Overview of stress variable calculations with <i>GeoStress</i> page <i>Stress</i>	24

List of Figures

Fig. 2.1-1:	Geometry, mesh and displacement boundary conditions of the example model.	9
Fig. 2.2-1:	Graphical user interface of Tecplot 360 EX.	11
Fig. 2.2-2:	Relation between <i>zone</i> , <i>zone set</i> , <i>time strand</i> and <i>solution time</i>	11
Fig. 2.2-3:	Creation of a cross section through the model.....	12
Fig. 2.2-4:	Creation of a new <i>zone set</i> and editing its <i>time strand</i> value.....	12
Fig. 2.4-1:	Dialog box for saving the results in the Tecplot *. <i>plt</i> -file format.	14
Fig. 2.4-2:	Dialog box for saving model results as a figure.	15
Fig. 2.4-3:	Dialog boxes for saving model results as a movie.....	16
Fig. 2.4-4:	Dialog boxes to save selected model results in ASCII data format.....	17
Fig. 2.6-1:	Loading Abaqus odb-files.....	18
Fig. 2.6-2:	Advanced loading options for Abaqus odb-files.	18
Fig. 3.1-1:	Page <i>General</i>	19
Fig. 3.1-2:	Definition of the geo-orientation.	20

Fig. 3.1-3:	Plot scaling factor setting in Tecplot 306 EX.....	21
Fig. 3.1-4:	Calculation of the variable True Vertical Depth <i>TVD</i>	22
Fig. 3.2-1:	Page <i>Stress</i>	23
Fig. 3.2-2:	Contour plot of the horizontal stress difference $S_{Hmax} - S_{Hmin}$	25
Fig. 3.2-3:	Contour plot of the Regime Stress Ratio (RSR).	25
Fig. 3.3-1:	Coulomb failure in a 2D plane.....	26
Fig. 3.3-2:	Page <i>Fracture Potential</i>	28
Fig. 3.3-3:	Calculation of the Fracture Potential with homogeneous rock properties.	28
Fig. 3.3-4:	Visualization of the Material ID number.	29
Fig. 3.3-5:	Plot of the Fracture Potential with inhomogeneous rock properties.....	29
Fig. 3.4-1:	Page <i>Coulomb Failure Stress</i>	30
Fig. 3.4-2:	Coulomb failure stress calculation.....	32
Fig. 3.4-3:	Single differential Coulomb failure stress between all model steps.	33
Fig. 3.4-4:	Cumulative differential Coulomb failure stress between model steps.....	34
Fig. 3.5-1:	Page <i>Slip and Dilation tendency</i>	36
Fig. 3.5-2:	Slip tendency calculation.....	37
Fig. 3.5-3:	Differential slip tendency between all zones.	38
Fig. 3.5-4:	Cumulative differential slip tendency between the user defined zones.	39
Fig. 3.6-1:	Page <i>Internal Surfaces</i>	40
Fig. 3.6-2:	Normal stress distribution on an internal surface.	43
Fig. 3.6-3:	Dip slip distribution on an internal surface.....	43
Fig. 3.7-1:	Page <i>External Surfaces/Polylines</i>	44
Fig. 3.7-2:	Shear stress distribution on an external surface.....	47
Fig. 3.7-2:	Vertical stress along an external polyline.....	48
Fig. 4.1-1:	Data set information dialog box.	50
Fig. 4.2-1:	Write Data as Text File dialog box.....	51
Fig. 4.3-1:	Write Data as Text File dialog box.....	52
Fig. 4.4-1:	Dialog box for calculation of variables.....	52
Fig. 4.4-2:	Visualization of the normal vector of surfaces.	53

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/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 in the course of 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 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).

In order 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 for the commercial visualization software Tecplot 360 EX (in the following we always refer to the version 2015 R2). 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 also 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 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 divided in 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 add-on *GeoStress* description and the accompanying application examples. For further details we refer to the manuals of Tecplot 360 EX. In this chapter 2 we also describe the installation of the add-on *GeoStress*. However, 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) or Hergert et al. (2015).

Chapter 3 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. Details of the NL model setup are presented in the NAGRA report NAB 13-88 (Heidbach

et al., 2013). Finally, chapter four presents a number of further Tecplot 360 EX applications in a geoscientific context that can potentially accelerate the analysis procedure significantly.

1.2 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.

Since there is no strict definition of stress terms (e.g. Engelder, 1992; 1994; Jaeger et al., 2007; Schmitt et al., 2012), a brief definition of the stress terms used in this report is given.

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 in-situ 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). The three remaining non-zero components in the diagonal of the matrix are the principal stresses where S_1 is the largest and S_3 the smallest ($S_1 \geq S_2 \geq S_3$).
differential stress	$S_d = S_1 - S_3$	Difference between the largest and the smallest principal stress.
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. Only at the Earth surface S_v is one of the principal stresses; at greater depth S_v can deviate from a 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.
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 (e.g. uniaxial, lithostatic). In particular a reference stress state implies that $S_H = S_h$. However, definitions in the literature are not consistent. Tectonic stress must not necessarily equal the deviatoric stress which is the non-isotropic part of the stress tensor (the isotropic part is the mean stress S_m or one third of the trace of the of stress tensor $S_m = 1/3(S_{xx} + S_{yy} + S_{zz}) = 1/3(S_1 + S_2 + S_3)$). This is only true when one assumes that the reference stress state is lithostatic ($S_{Hmax} = S_{hmin} = S_v$ with the assumption that S_v is a principal stress).
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 (C��lerier, 1995). Normal faulting: $S_v > S_{Hmax} > S_{hmin}$; Thrust faulting: $S_{Hmax} > S_{hmin} > S_v$; Strike-slip: $S_{Hmax} > S_v > S_{hmin}$.

2 Data visualisation with Tecplot 360 EX

2.1 Introduction

The following explanations are accompanied with an example using a geomechanical-numerical model of a potential siting area for high-level radioactive waste in Northern Switzerland. The model has a number of 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 in order 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.

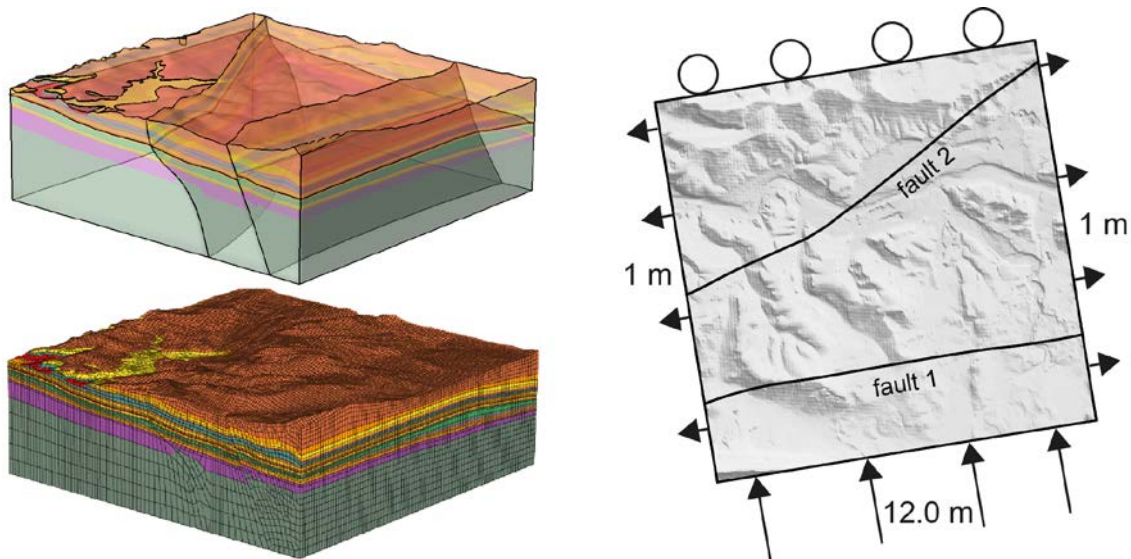


Fig. 2.1-1: Geometry, mesh and displacement boundary conditions of the example model.

Upper left figure shows the layering of the lithology and the location of the two faults that cut the model into three blocks and the 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 of this example are provided in the file *example.odb*. The Tecplot-file is named accordingly *example.lay* and *example.plt*. All other additional files needed to perform with the add-on *GeoStress* the model result visualization are described in the respective section.

2.2 Basic concept and terminology of Tecplot 360 EX

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 a number of 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 zones 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 *zone sets*. Each *zone set* contains several *zones*. *Zones* can be grouped to a *zone set* by assigning them to the same *time strand* value which is automatically done when you load your Abaqus odb-file. E.g. the *example.odb* has four increments that are assigned to the same *time strand* and displayed in chronological consecutive order according to their assigned solution time and thus the four increments are grouped in one *zone set* with four *zones*. Note that the value of the *time strand* is not related to the modelling time which is stored in the variable *solution time*. In this sense the *time strand* is an identifier that controls which *zone* (e.g. all increments of the model) are grouped into a *zone set*. When e.g. a cross section is generated through the 3D model volume all variables are interpolated onto the cross section and stored in a new *zone set* with a new *time strand* number. The new *zone set* given in the example consists of four *zones*.

Fig. 2.2-3 and Fig. 2.2-4 show how cross sections over time are extracted. For this procedure enable the *Slices* option from the side bar (turn off *Contour* and *Mesh* in the side bar. Then choose from the menu bar the option *Data/Extract/Extract Slices Over Time* (Fig. 2.2-3). With the *Zone Style...* dialog from the side bar the first *zone set* (*zones* 1-4) can be turned off (Fig. 2.2-3). 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 *zone set* contains all *zones* of the model. Again each chronological and geometrical entity in each step of the Abaqus analysis is one *zone*.

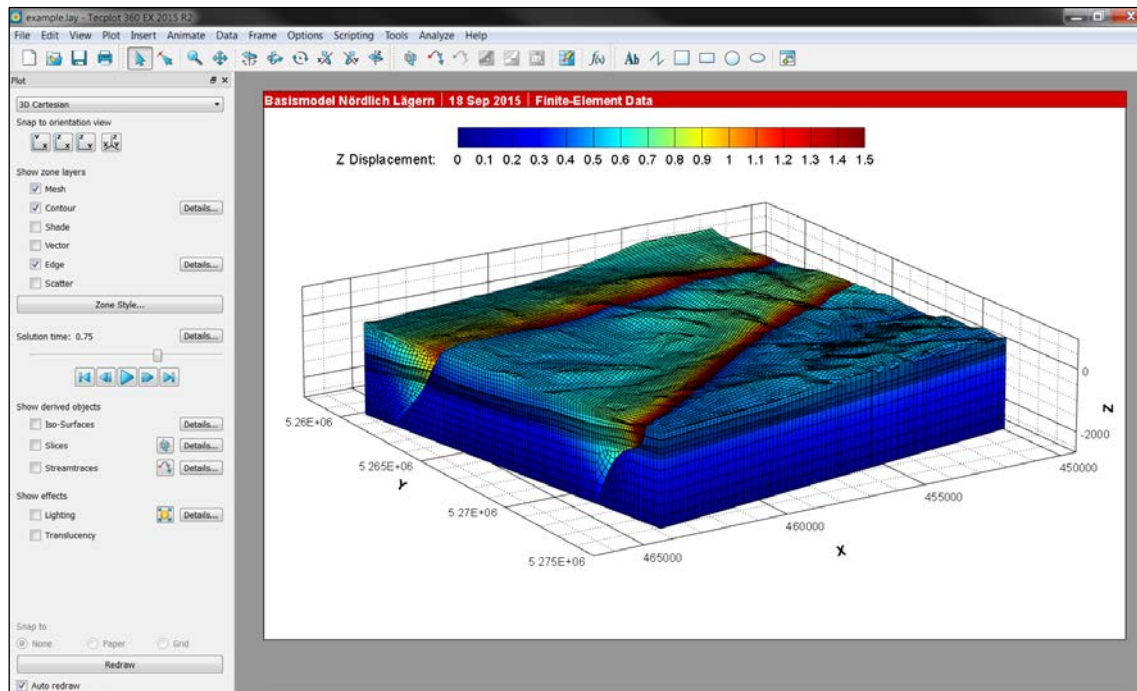


Fig. 2.2-1: Graphical user interface of Tecplot 360 EX.

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.

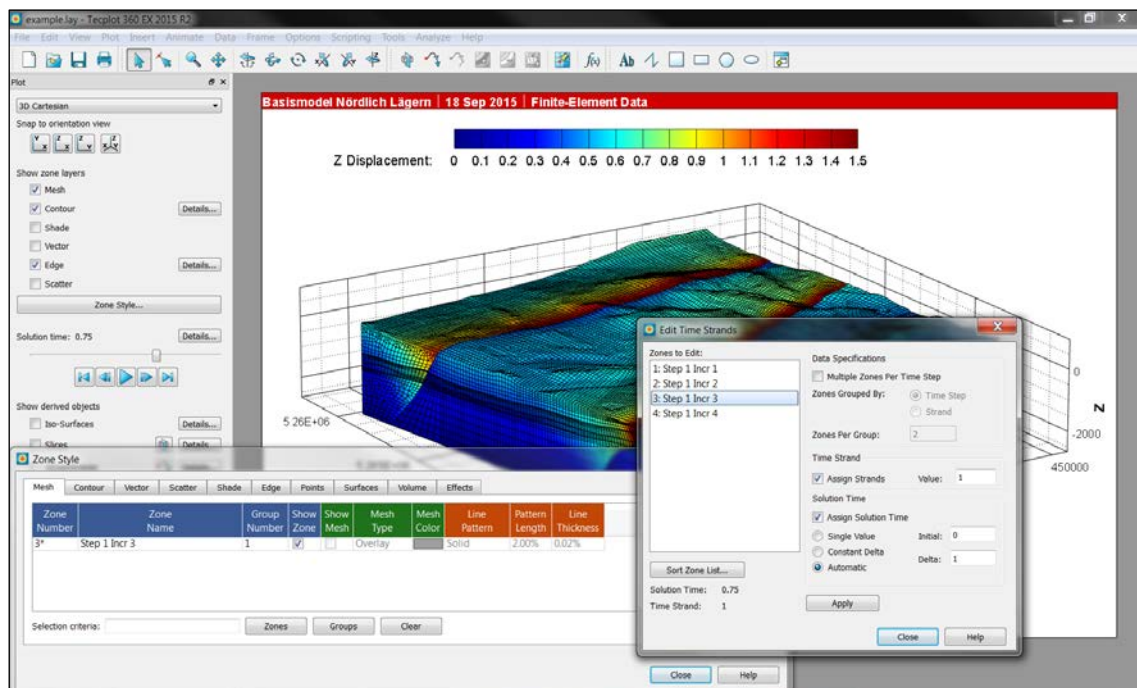


Fig. 2.2-2: Relation between *zone*, *zone set*, *time strand* and *solution time*.

Each time increment of the Abaqus analysis is in Tecplot one *zone*. Several *zones* can be grouped into one *zone set* 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.

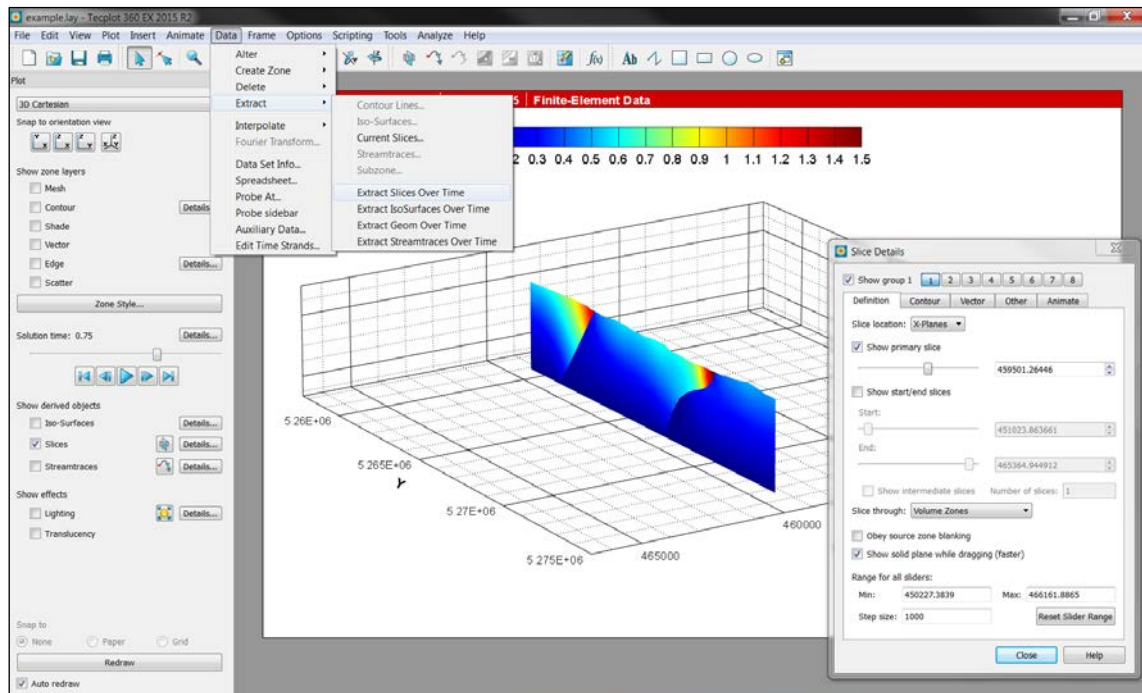


Fig. 2.2-3: Creation of a cross section through the model.

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

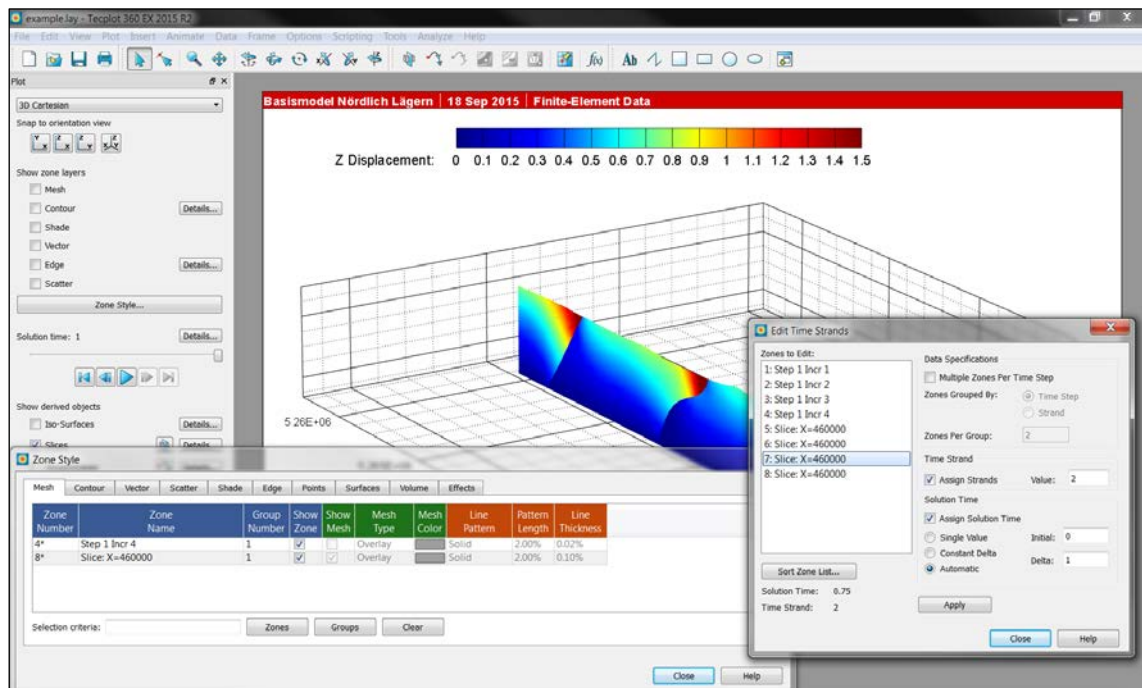


Fig. 2.2-4: Creation of a new zone set and editing its time strand value.

Open the *Zone Style...* in the side bar and turn off the first zone set (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 zone set with a new (here 2) time strand value.

2.3 The Add-on *GeoStress*

The add-on *GeoStress* detects the main geometry data of the model (total number of nodes and elements) from the first *zone*. You can also load a second model, i.e. a second model data set into Tecplot 360 EX, but the add-on *GeoStress* would not recognize this second new set.

GeoStress needs at minimum 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 (SXX, SYY, SZZ, SXY, SYZ, SZX) 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 the 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. Thus, Tecplot 360 EX (release 2015 R2) usually cannot load the result files from the latest version of the commercial software packages. E.g. for Abaqus the highest version of the odb-file that can be loaded is v6.11.

The add-on *GeoStress* either adds further variables to the existing first *zone set* or it adds new *zone sets* to the model data set and interpolates all variables to the new *zone set*. 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 package output file is not supported).
2. Generate with *GeoStress* all new variables in the first *zone set*. Note that new or updated variables using *GeoStress* are not updated in *zone sets* (e.g. cross sections or external surfaces) that were generated before that update/new generation.
3. Generate new *zone sets* (e.g. cross sections or external surfaces) either with *GeoStress* or using the functionality of Tecplot 360 EX.

2.4 Saving Tecplot 360 EX format and output formats

2.4.1 Tecplot file formats *.lay and *.plt

To save the current view settings of all frames the menu bar offers to save the layout with *Save Layout* or *Save Layout As...* The *.lay-file saves the current view settings, but the time consuming loading procedure of the *.odb-file has to be performed again. The *.lay-file saves only the plot view settings of Tecplot 360 EX, but not the re-structuring of the data from the *.odb-file.

In order to reduce the loading time of the model result file the loaded data can be saved in the binary *.plt-file format of Tecplot 360 EX. 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. 2.4-1 the user can control which Abaqus steps, variables and other information from the binary *.odb-file will be stored in the *.plt-file format. The *.lay-file is then linked to the *.plt-file.

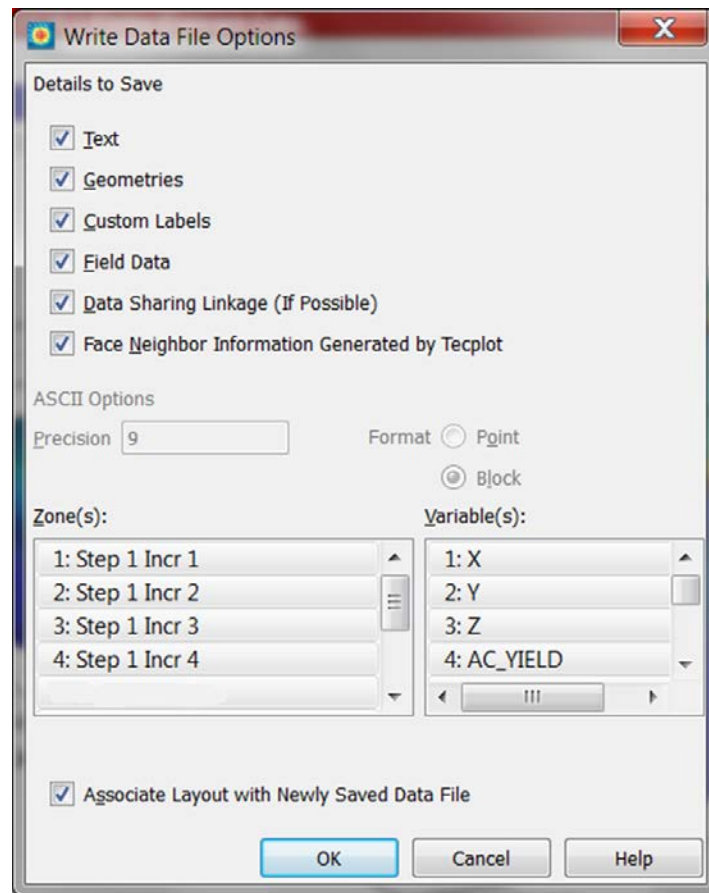


Fig. 2.4-1: Dialog box for saving the results in the Tecplot *.plt-file format.

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.

2.4.2 Figure formats

Saving the active frame as a figure the option *File/Export...* from the menu bar is used. Here the user can choose between three 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. 2.4-2). 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 is one vector.

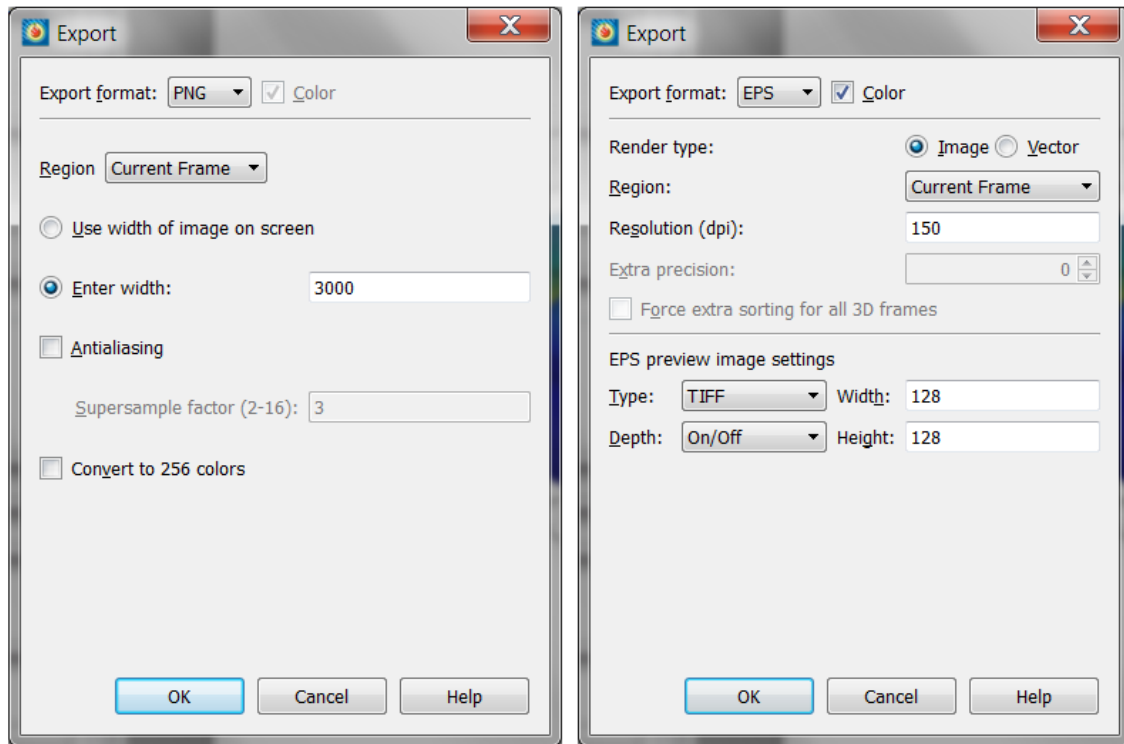


Fig. 2.4-2: Dialog box for saving model results as a figure.

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.

2.4.3 Movie formats

The model results can also be saved as movies in different formats (Fig. 2.4-3). 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 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. 2.4-3 on the left and the centre dialog boxes. When *Destination/To file* is selected on the left dialog box the dialog box in the centre of Fig. 2.4-3 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 active frame within the dialog box on right of Fig. 2.3-3. Tecplot 360 EX interpolates between the different frames and generates a movie that is either saved in a file or displayed on the screen.

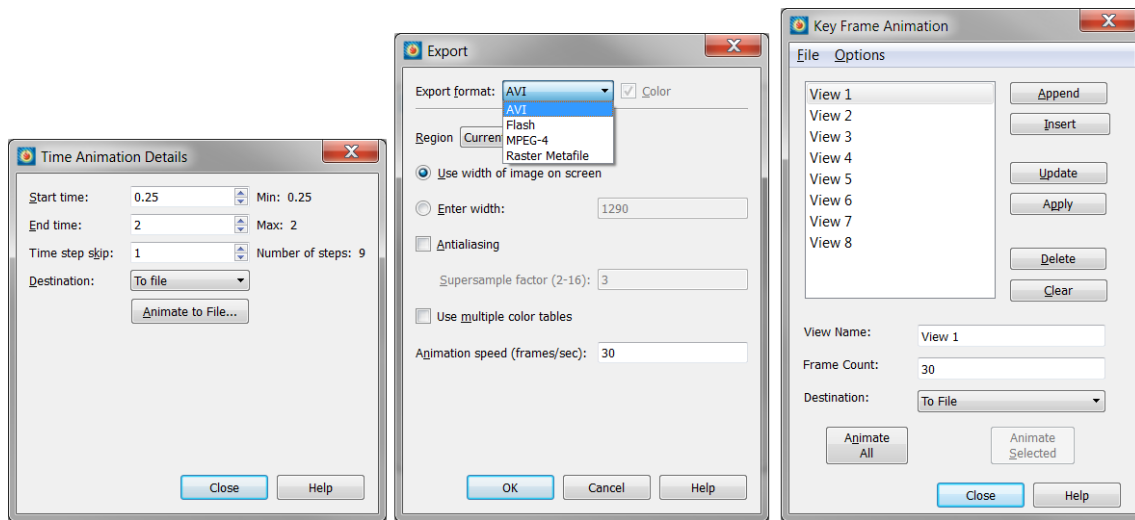


Fig. 2.4-3: Dialog boxes for saving model results as a movie.

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.

2.4.4 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. 2.4-4) the user can choose from which zone (e.g. a certain slice at a certain time step of the model; see Fig. 2.3-4) which variable is written to the ASCII file. To create a line-wise output of the data choose the *Format Point* toggle.

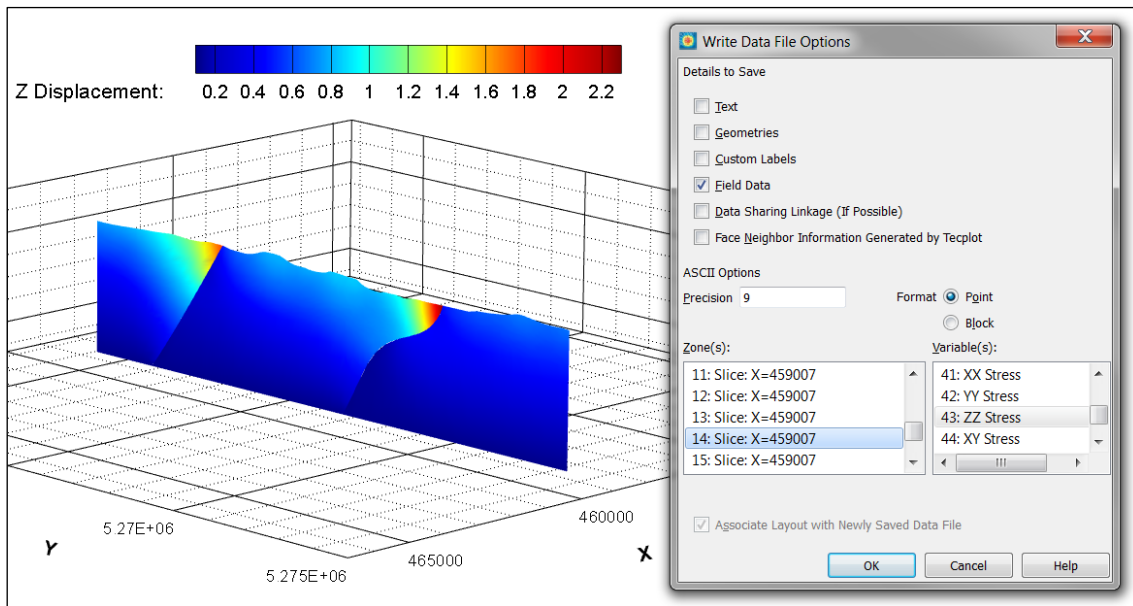


Fig. 2.4-4: Dialog boxes to save selected model results in ASCII data format.

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 Installation of the Tecplot 360 EX add-on *GeoStress*

The add-on *GeoStress* is a shared library compiled for a 64-bit Windows platform (*GeoStress_win64.dll*) and a Linux platform (*libGeoStress_lin64.so*) for the Tecplot 360 release 2015 EX R2. In order 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 have to be added at the end of the file:

```
$!LoadAddon "GeoStress_win64
```

for Windows and

```
$!LoadAddon "GeoStress_lin64"
```

(without prefix "lib") for Linux. When you start Tecplot 360 EX you will find the new add-on *GeoStress* is available under the menu bar *Extras*. There are other useful add-ons provided in the Internet platform Tecplot Talk (www.tecplottalk.com). The installation is similar to the one described above. However, most of these add-ons are not yet compiled for the version Tecplot 360 EX 2015 R2 that was released in July 2015.

2.6 Loading of the Abaqus odb- and inp-files

Tecplot can load three different Abaqus 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. Tecplot 360 EX can also load the output files from other commercial finite element software packages such as Nastran or Ansys. In principal the add-on *GeoStress* should work with all kind of finite element output files, but it has not been tested. 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 (S_{XX} , S_{YY} , S_{ZZ} , S_{XY} , S_{XZ} , S_{YZ}) and the three components of the displacement vector (X displacement, Y displacement, Z displacement). Note that *GeoStress* cannot be used for 2D model results since most calculations are based on the 3D stress tensor.

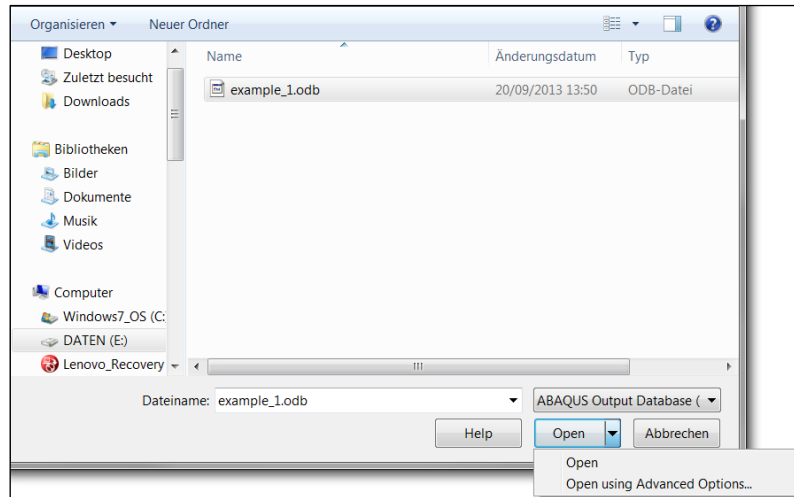


Fig. 2.6-1: Loading Abaqus odb-files.

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

The dialog box for loading odb-files Tecplot 360 EX offers to use advanced options (Fig. 2.3-1). However, the option *Subdivide Zones by Components* or *by Elements Type* can lead to limits when the add-on *GeoStress* is used. 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 *zone set* of the model data set (see also section 3.1 for further details).

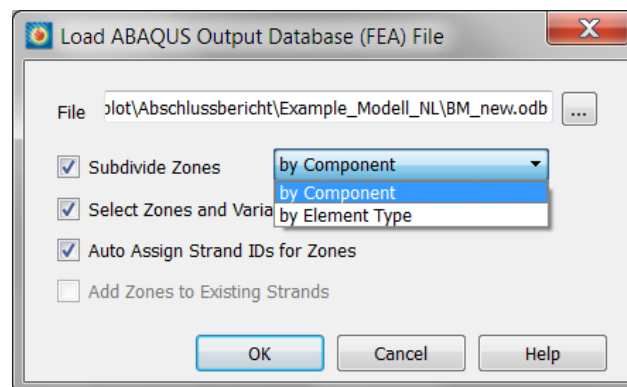


Fig. 2.6-2: Advanced loading options for Abaqus odb-files.

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

3 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. 3.1-1). 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 3.1-3.7 of this chapter. In the first five tab cards new variables are calculated and provided for the first zone set. Only the options in page six and seven generate both new variables and new zones (set of zones) on which all variables are interpolated.

3.1 General settings

The first page of the add-on *GeoStress* is the General Setting. The page has five sections (Fig. 3.1-1). In particular the setting of the stress and plot scaling values is important as this has an impact on the variable calculation in the other tab cards.

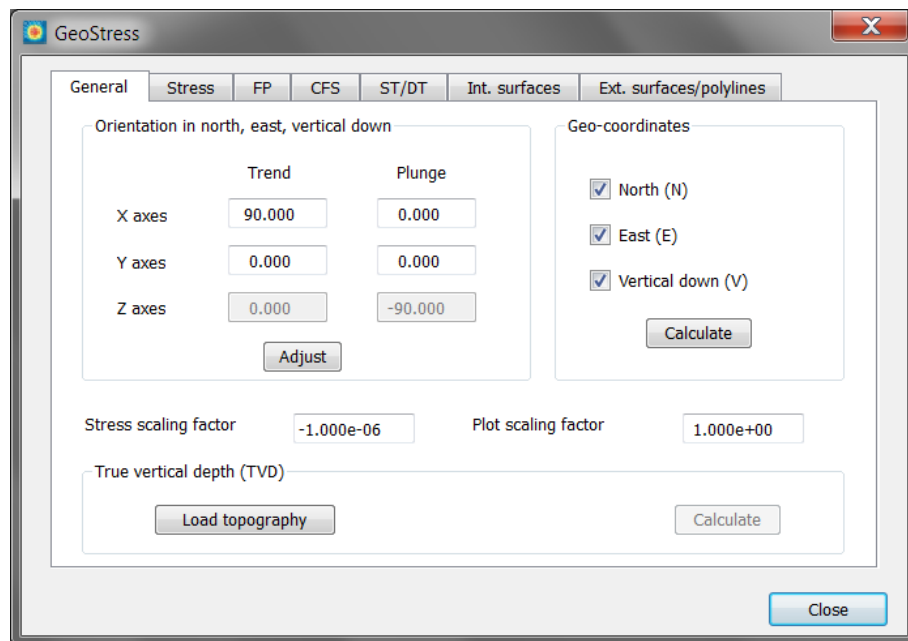


Fig. 3.1-1: Page *General*.

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).

3.1.1 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 $\{N \text{ (north)}, E \text{ (east)}, V \text{ (vertical down)}\}$. 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. 3.1-2).

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 -axis is then determined automatically by *GeoStress* because it is over-determined 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.

3.1.2 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.

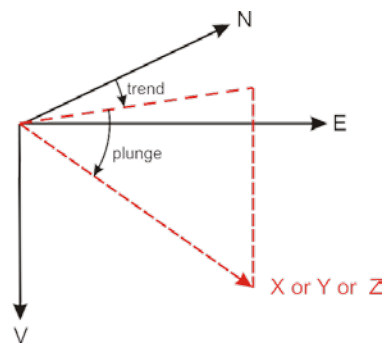


Fig. 3.1-2: Definition of the geo-orientation.

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.

3.1.3 Stress scaling factor

In contrast to the geosciences community the commercial finite element software packages define compression 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). 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/Dilation Tendency (*ST/DT*) are expected in the same unit as defined in the general setting of the add-on.

3.1.4 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. In the appearing dialog box (Fig. 3.1-3) the tick box *Deform Plots by Factor* changes the plot scaling factor; the de-

fault value of Tecplot 360 EX as well as for the add-on *GeoStress* is 1. 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. The user has to make sure that the finally stored plot scaling factor that is used from Tecplot 360 EX is consistent with the post scaling factor to be given in the *GeoStress* add-on in the first page with the general setting. If this consistency is not given the calculation of the new variable *True Vertical Depth (TVD)* and the kinematic estimates on the faults would be wrong. This is necessary since *TVD* is calculated for each step and therefore a variable that changes over time due to the deformation of the geometry.

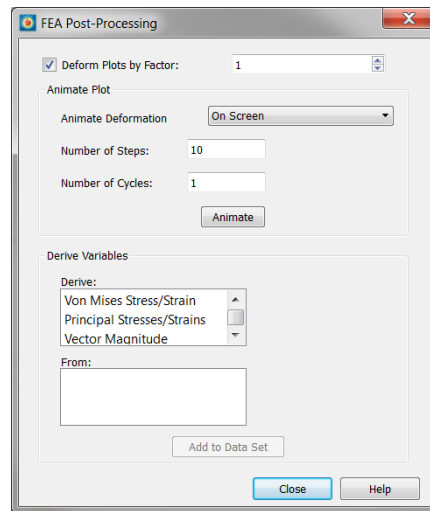


Fig. 3.1-3: Plot scaling factor setting in Tecplot 306 EX.

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.

3.1.5 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 takes into account 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:

```
**_ABQUS_Input_Deck_Generated_by_HyperMesh_Version_12.0.0.85
**_Generated_using_HyperMesh-Abaqus_Template_Version_12.0
**
*NODE
100239, 681815.55544868, 268163.90825397, 407.77080517579
100243, 684885.07511532, 268705.14852239, 363.795577031
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, *Geo-Stress* 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.

3.1.6 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 general page of input (Fig. 3.1-1). The calculation is performed for each increment of the model taking into account also the changes of the *TVD* with time. For high-resolution models with a large number of nodes at the model surface (i.e. the topography nodes) and lots of increments the *TVD* calculation can be quite time-consuming.

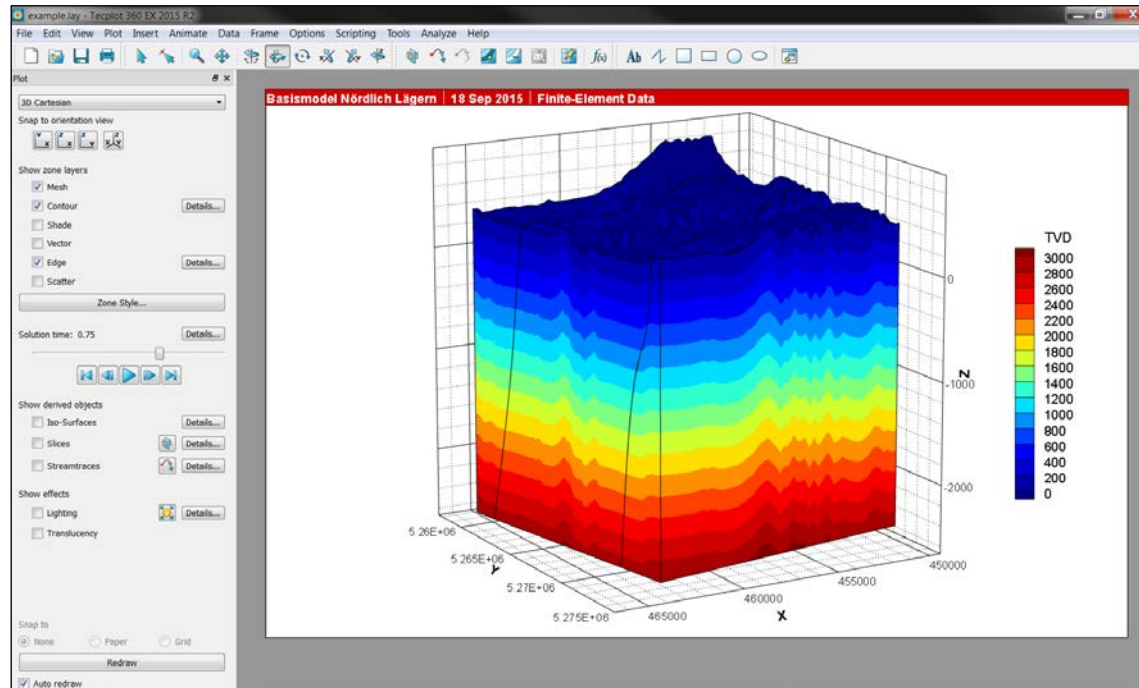


Fig. 3.1-4: Calculation of the variable True Vertical Depth *TVD*.

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

3.2 Stress value calculations

3.2.1 Theory and background information

The page *Stress* (Fig. 3.2-1) offers the calculation of a wide range of stress variables that are derived from the six components of the stress tensor. An overview as well as details and explanation of the different variables that are calculated on this page are summarized in Table 3.2-1. The page is divided in two parts. The upper area provides all information on the principal

stresses S_1 , S_2 and S_3 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. 2.3-3 and Fig. 2.3-4 in section 2.3) 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. Note that new variables that are calculated with the equations option are not stored in the plt-file, but only on the lay-file. 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 new zones.

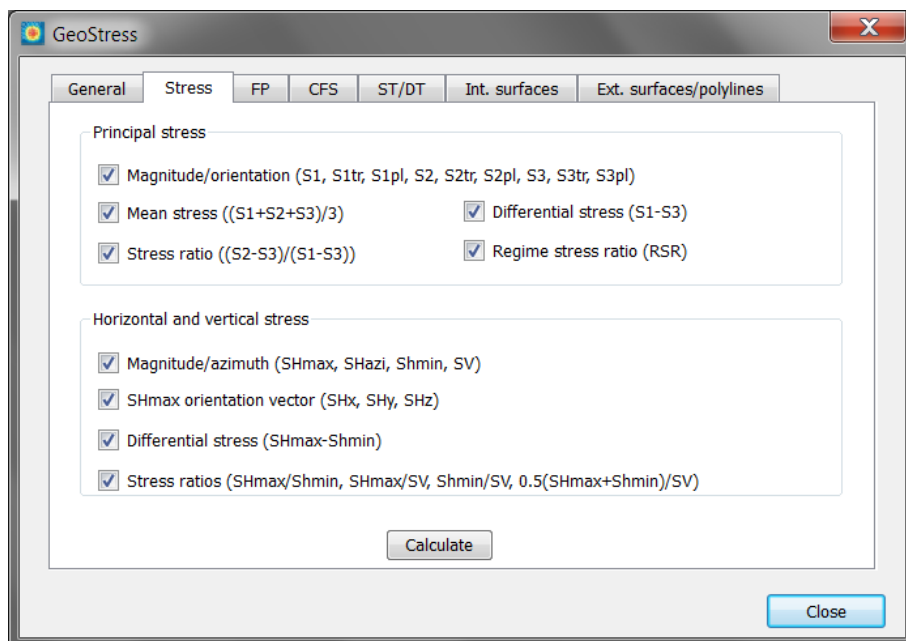


Fig. 3.2-1: Page *Stress*.

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.

Tab. 3.2-1: 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.

Variable	Definition/Comment
$S1, S2, S3$	Magnitudes of the three principal stresses (the stress unit depends on the choice in the general settings of the first page of <i>GeoStress</i>).
$S1tr, S1pl$ $S2tr, S2pl$ $S3tr, S3pl$	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 <i>GeoStress</i> (see Fig. 3.1.-2).
$(S1+S2+S3)/3$	Mean stress. Note that $(S1+S2+S3)/3 = (S_{xx}+S_{yy}+S_{zz})/3$ since this is one of the three invariants of the stress tensor.
$S1-S3$	Differential stress as the difference between the largest ($S1$) and the smallest principal stress ($S3$).
$(S2-S3)/(S1-S3)$	Stress ratio. When $S1-S3=0$ the stress state is lithostatic (sometimes also called isotropic) and $(S2-S3)/(S1-S3)=1$.
RSR	<p>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</p> $RSR = (n + 0.5) + (-1)^n (SR - 0.5) \quad n = \begin{cases} n = 1 & \text{if } Shmin \leq SV \leq SHmax \\ n = 2 & \text{if } SV \leq Shmin \leq SHmax \\ n = 0 & \text{otherwise} \end{cases}$ <p>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 ($RSR=2$).</p>
$SHmax, SHazi$ $Shmin, SV$	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 vector of the $SHmax$ orientation $SHazi$ with respect to the Cartesian coordinate system $\{X, Y, Z\}$. Note that SHz should be zero and that only the two components SHx and SHy are needed to plot with Tecplot 360 EX the $SHmax$ 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 $SHmax/Shmin=-999$, if $SV=0$ the other three ratios are set to -999.

3.2.2 Application examples

The following two figures show the results of the horizontal stress difference $S_{Hmax}-S_{Hmin}$ (Fig. 3.2-1) and the Regime Stress Ratio RSR (Fig. 3.2-2), respectively.

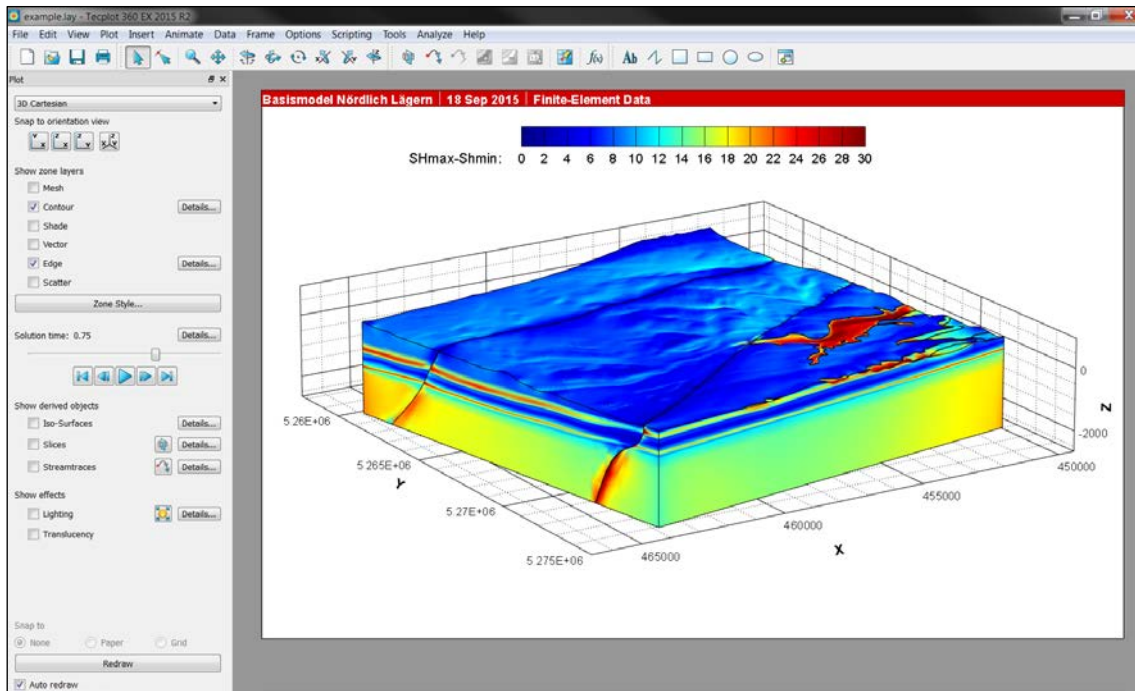


Fig. 3.2-2: Contour plot of the horizontal stress difference $S_{Hmax} - S_{Hmin}$.

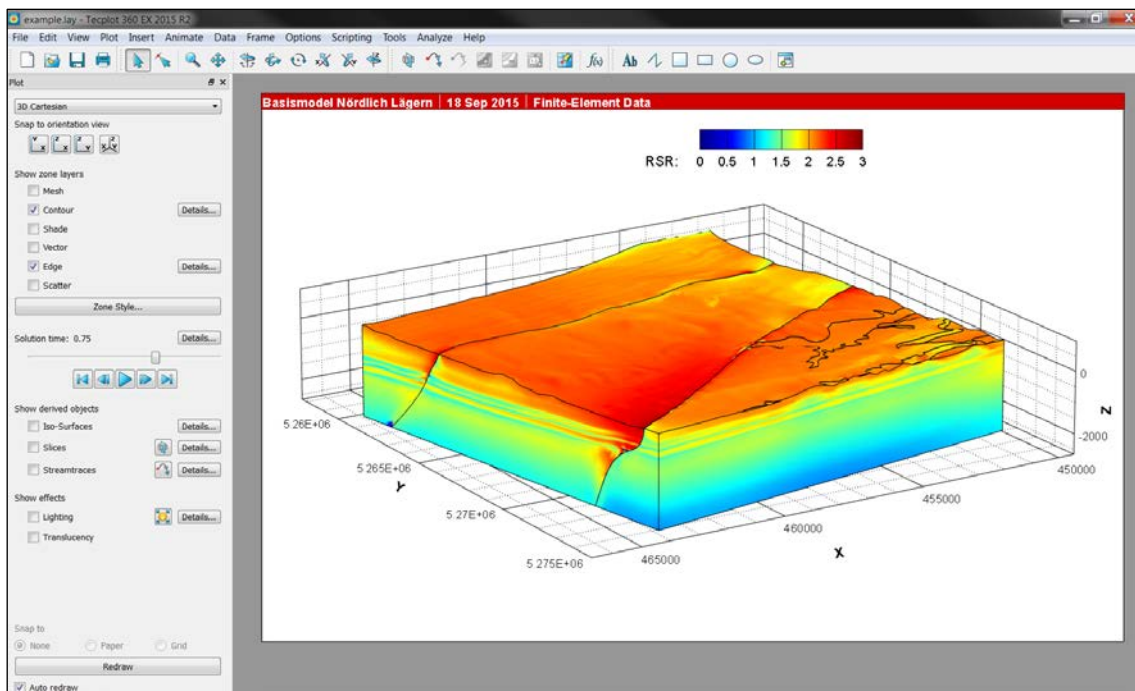


Fig. 3.2-3: Contour plot of the Regime Stress Ratio (RSR).

3.3 Fracture potential

3.3.1 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. 3.1-1). In particular *FP* is a relative measure for the Coulomb shear failure criterion in a volume with a given stress state:

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

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:

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

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

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

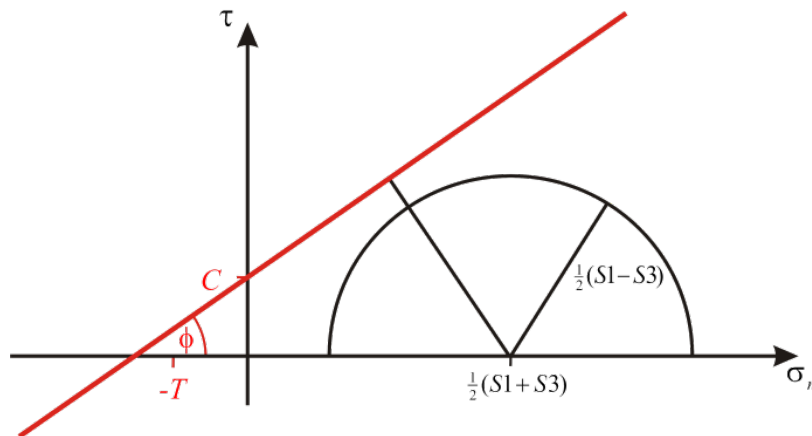


Fig. 3.3-1: Coulomb failure in a 2D plane.

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:

$$FP = \begin{cases} -999 & \text{if } \frac{1}{2}(S1 + S3) \leq -T & \text{(tensile fracture)} \\ -9999 & \text{if } -T < \frac{1}{2}(S1 + S3) \leq \frac{C}{\tan \phi} & \text{(undefined)} \\ \frac{1}{2}(S1 - S3)/(C \cos \phi + \frac{1}{2}(S1 + S3) \sin \phi) & \text{otherwise} \end{cases}$$

where T is the tensional strength of the rock. This value can be set in the *GeoStress* page for the *FP* calculations (Fig. 3.2-2). Again, the user of the add-on is responsible that the used units are consistent.

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 an element node are calculated as the mean value of 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:

```
*****
** 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       Wildegge 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 ****. In column 1 the *Material ID* is defined, in column 2 the *friction angle* ϕ , in column 3 the *cohesion* C and in 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.

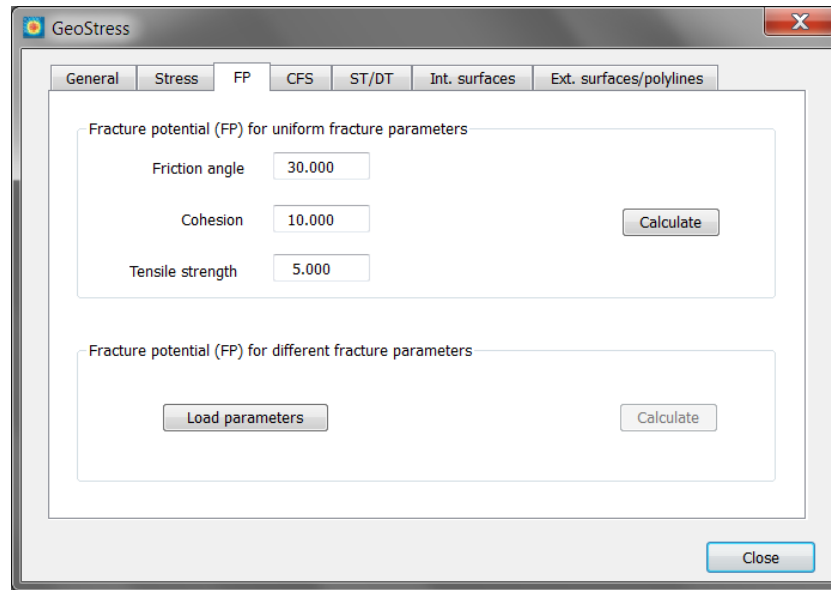


Fig. 3.3-2: Page *Fracture Potential*.

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

3.3.2 Application examples

Fig. 3.3-3 shows the calculation of the Fracture Potential with one cohesion and friction angle for the whole model volume. Fig. 3.3-4 displays the material ID number that is needed to create the file that connects the *Mat ID* with variable cohesion and friction angle for each individual geomechanical unit of the model. Then Fig. 3.3-5 shows the resulting *FP* values.

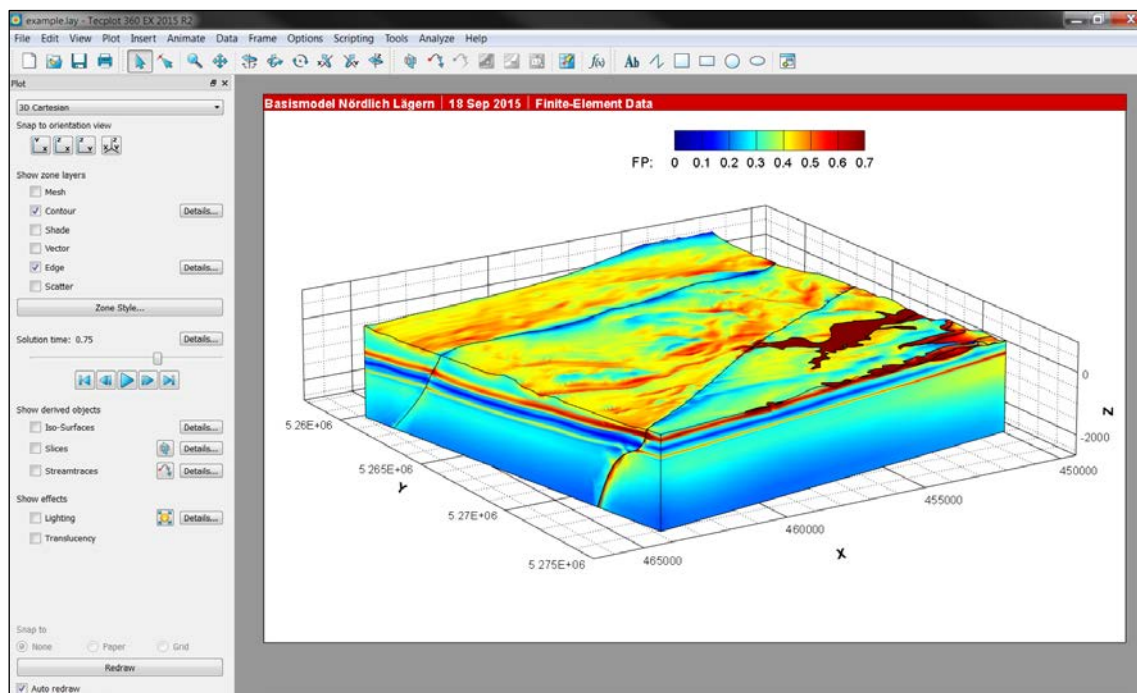


Fig. 3.3-3: Calculation of the Fracture Potential with homogeneous rock properties.

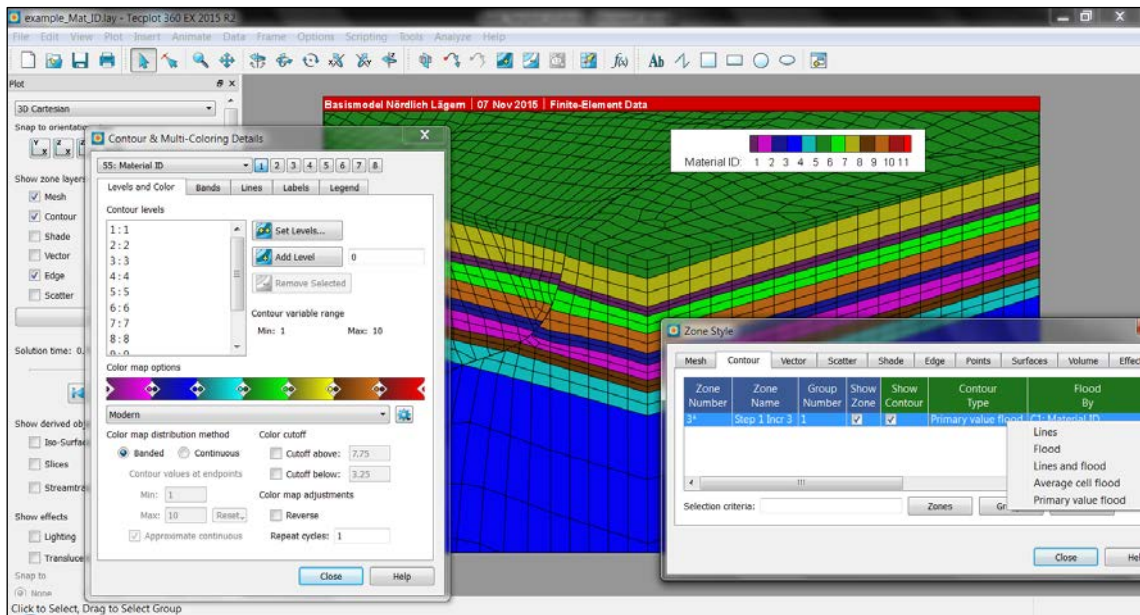


Fig. 3.3-4: Visualization of the Material ID number.

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

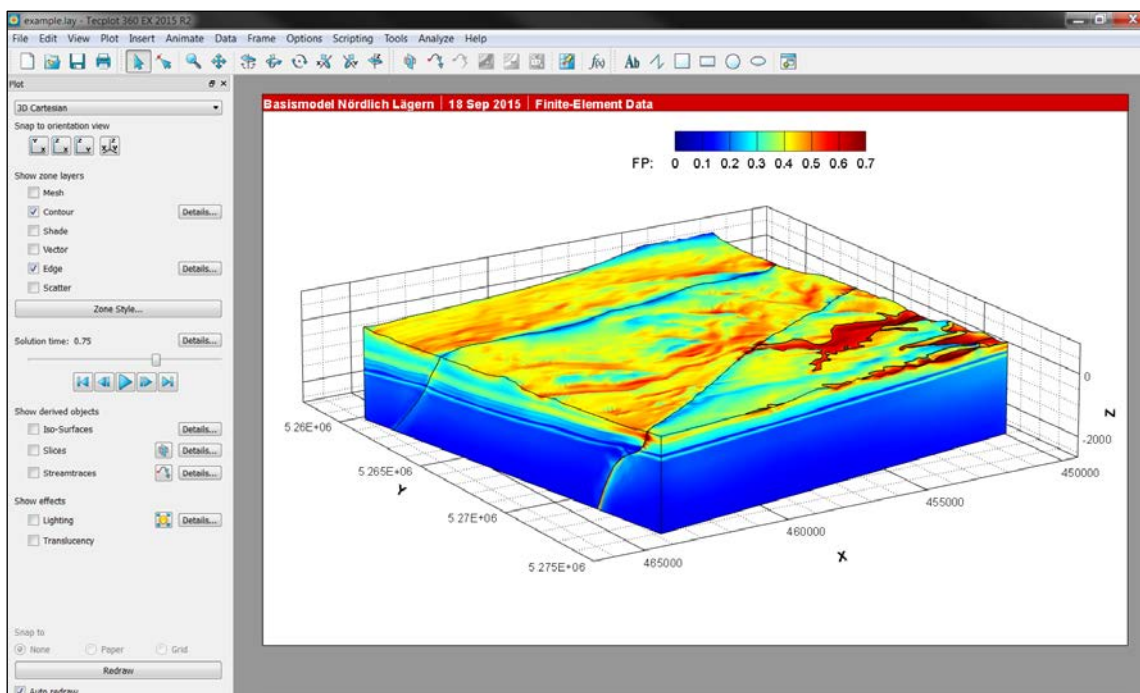


Fig. 3.3-5: Plot of the Fracture Potential with inhomogeneous rock properties.

Note the difference to the results displayed in Fig. 3.3-3 where a homogeneous rock property values for the cohesion and the friction angle is used.

3.4 Coulomb failure stress

3.4.1 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, $\tan(\phi)$ 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 priori selected rake direction; the latter is essential in order 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 the normal stresses become sufficiently negative the faults would not fail in shear (Mode II) but in dilation, i.e. opening (Mode I). Thus, *GeoStress* assigns *CFS*=-999 in the case when the normal stress σ_n is a tension (negative sign) and exceeds the given tensile strength T ($-\sigma_n \geq T$).

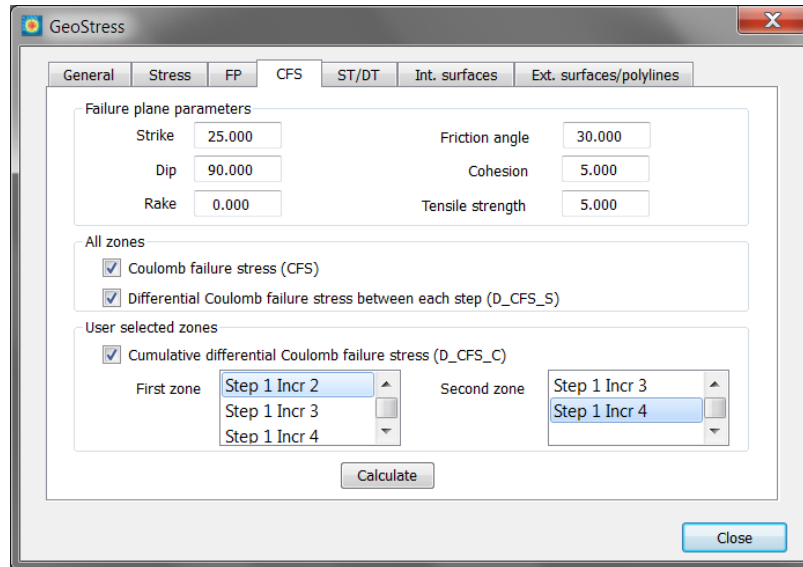


Fig. 3.4-1: Page *Coulomb Failure Stress*.

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 according to 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, $\pm 180^\circ$. 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 $\pm 180^\circ$ means the hanging wall moves toward the observer (right lateral motion). For any rake $> 0^\circ$ and $< 180^\circ$, the hanging wall moves up, indicating reverse motion on the fault; for any rake $< 0^\circ$ and $> -180^\circ$ 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. 3.4-1). 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 zones. *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 zones (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 increasing 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.

3.4.2 Application examples

Figure 3.4-2 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 3.4-3 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 3.4-4 shows the cumulative differential Coulomb failure stress *D_CFS_C* between two user selected zones. 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).

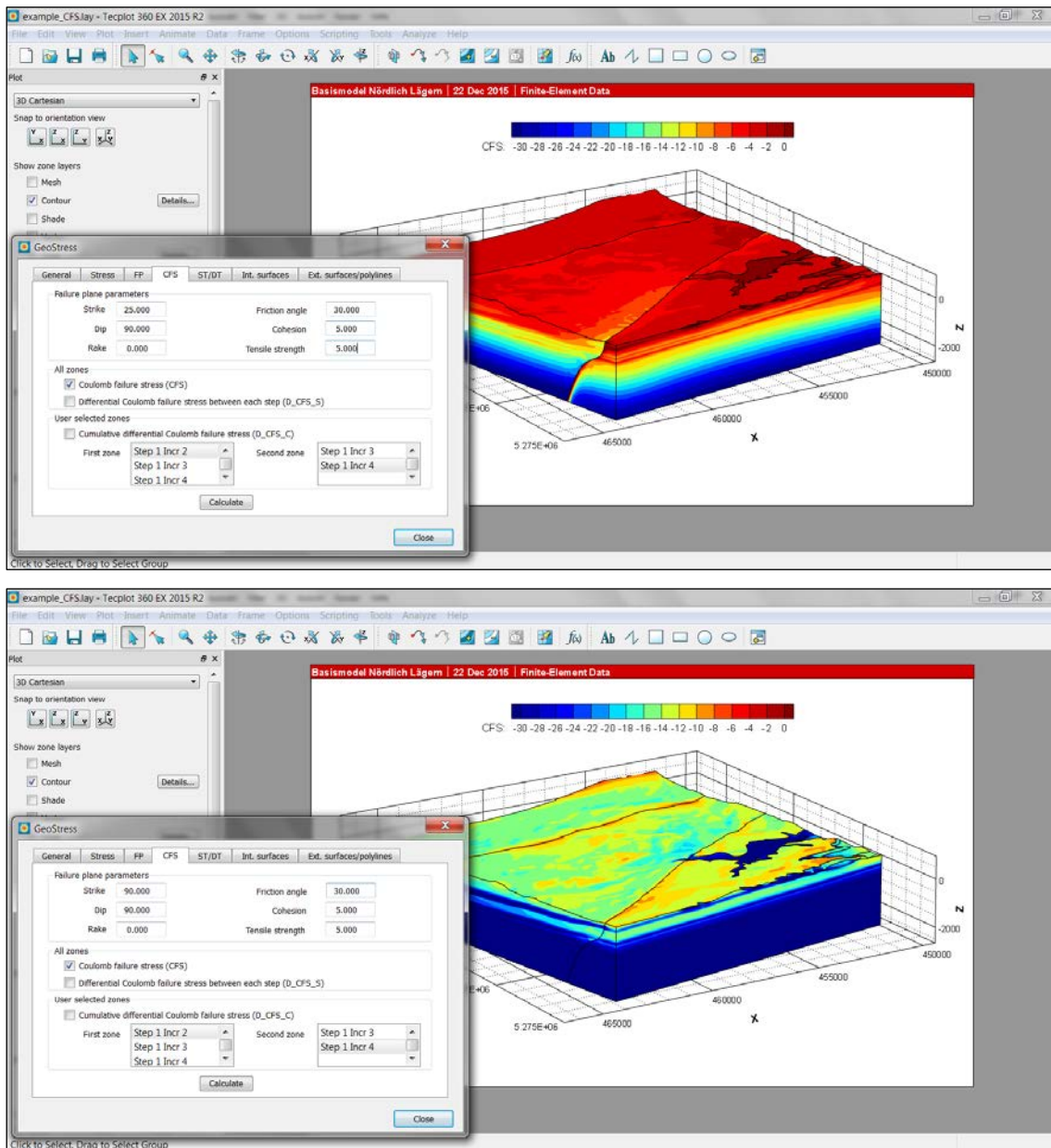


Fig. 3.4-2: Coulomb failure stress calculation.

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°

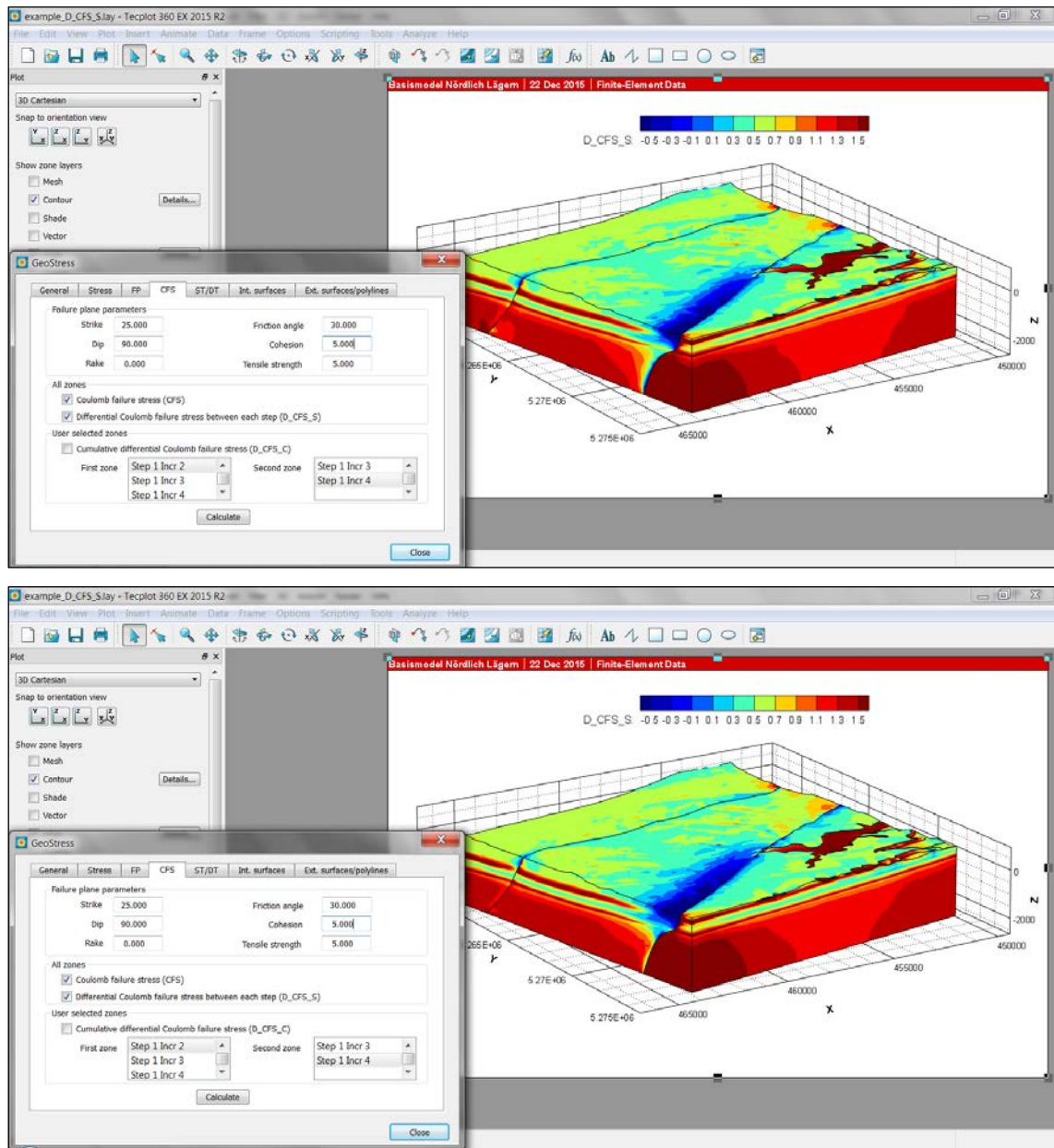


Fig. 3.4-3: Single differential Coulomb failure stress between all model steps.

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 Incr3. 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.

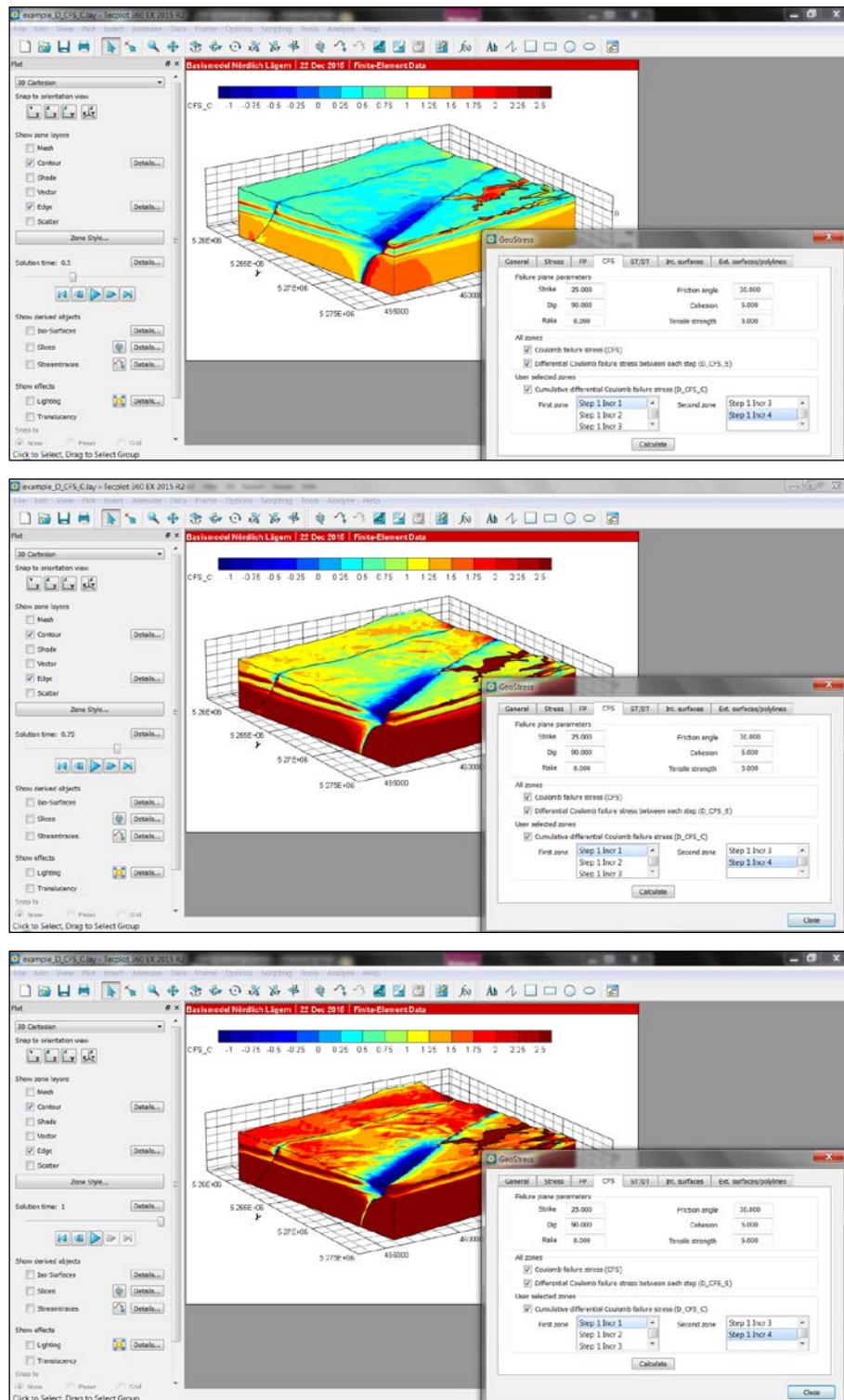


Fig. 3.4-4: Cumulative differential Coulomb failure stress between model steps.

The figures show the D_{CFS_C} in MPa for three steps. The upper figure shows D_{CFS_C} for Step1 Incr2 – Step 1 Incr1, the middle one for Step1 Incr3 - Step 1 Incr2 and the lower one for Step1 Incr4 - Step 1 Incr3. Thus the value of D_{CFS_C} that is presented in a zone is always the difference to the selected first zone.

3.5 Slip and dilation tendency

3.5.1 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 $\tan(\phi)$ where ϕ is the friction angle. Taking also into account the cohesion C the ST value can be defined as:

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

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 has to set $\phi = 45^\circ$ 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. 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 3.4. The only difference is that the rake information is not needed and neither the value of the tensile strength.

Calculation of ST and DT values

The calculation of ST/DT is divided into the sections *All zones* and *User selected zones* (Fig. 3.5-1). In the first section the variables are estimated for all zones (model increments/steps). Besides the absolute values for ST/DT the differential value between consecutive zones 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 zones (model steps/increments) the cumulative differential slip or dilatation tendency (D_ST_C , D_DT_C) is calculated. The D_ST_C/D_DT_C variables are zero for the selected first zone and then increasing from step to step until the second zone is reached. Positive differential ST/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/DT values (single and cumulative) indicate that the fault has been brought relatively further away from failure. Note that the ST/DT values are relative measures and that even an increase in ST/DT can occur when the absolute distance to the failure envelope has increased.

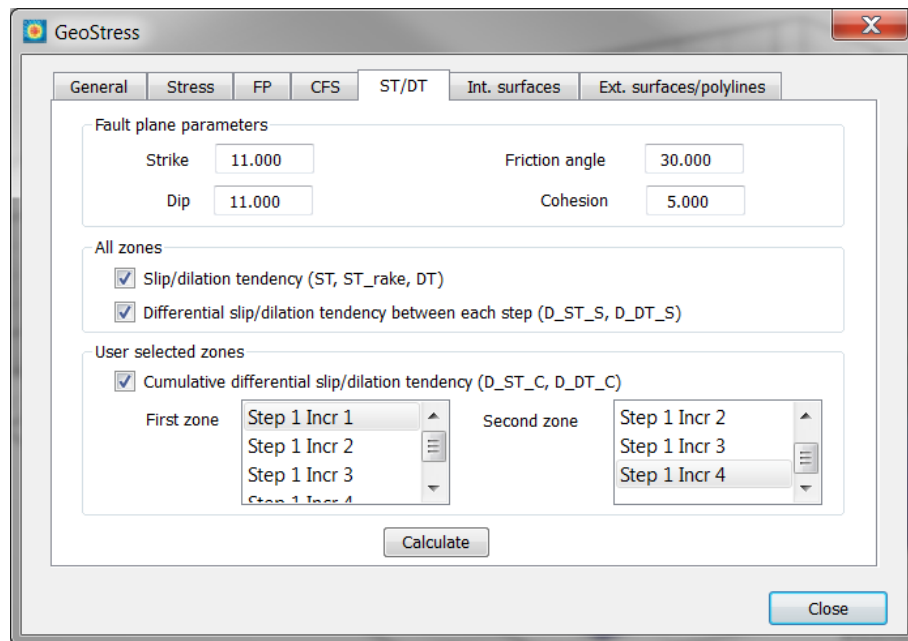


Fig. 3.5-1: Page *Slip and Dilation tendency*.

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/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/D_DT_S). In the lower section the cumulative difference of these values (D_ST_C and D_DT_C) between two zones selected by the user is calculated.

3.5.2 Application examples

Figure 3.5-2 shows the ST values as a contour plot. Both ST values are calculated for vertical faults with a strike of $25^\circ N$ 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.

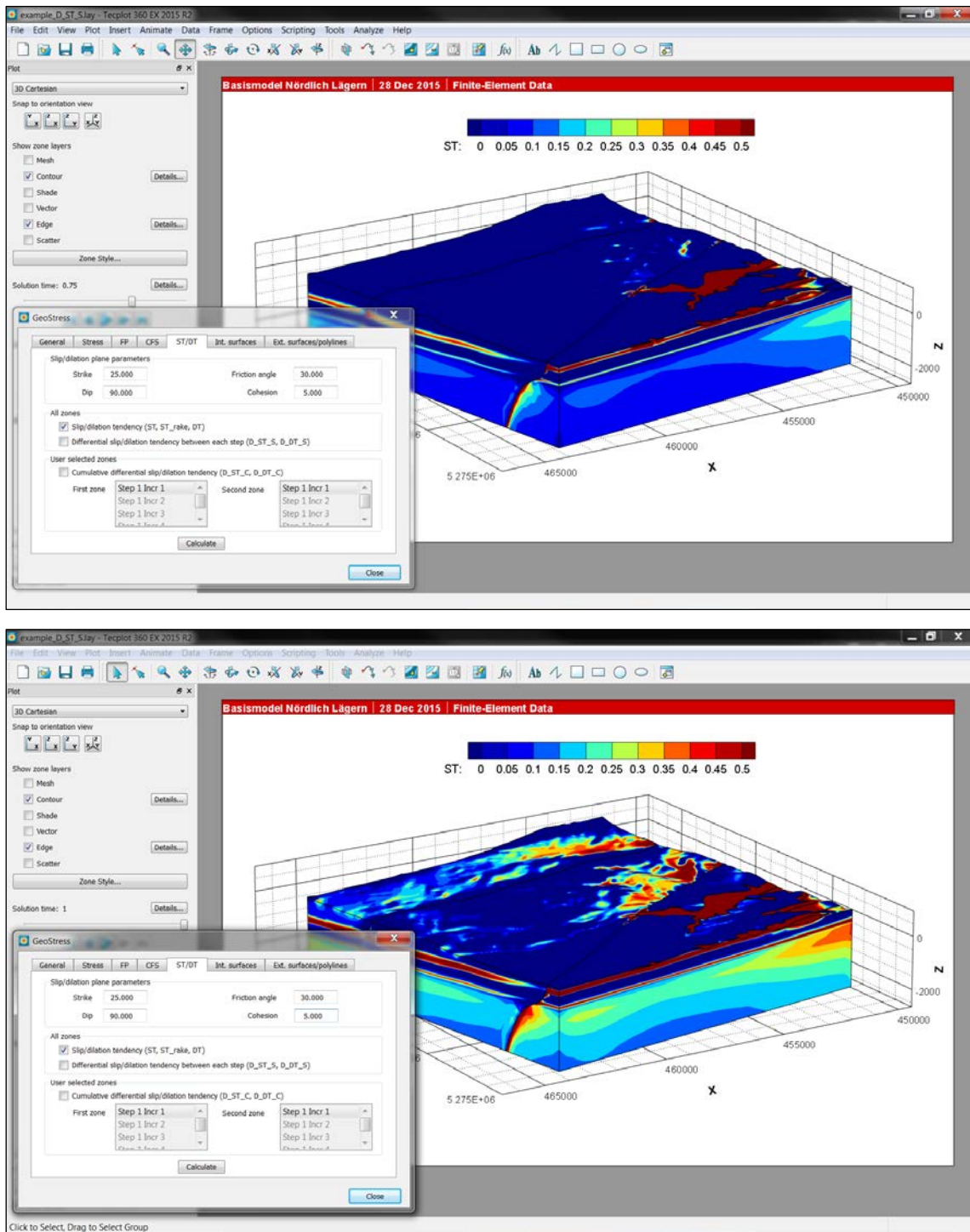


Fig. 3.5-2: Slip tendency calculation.

Both figures shows 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).

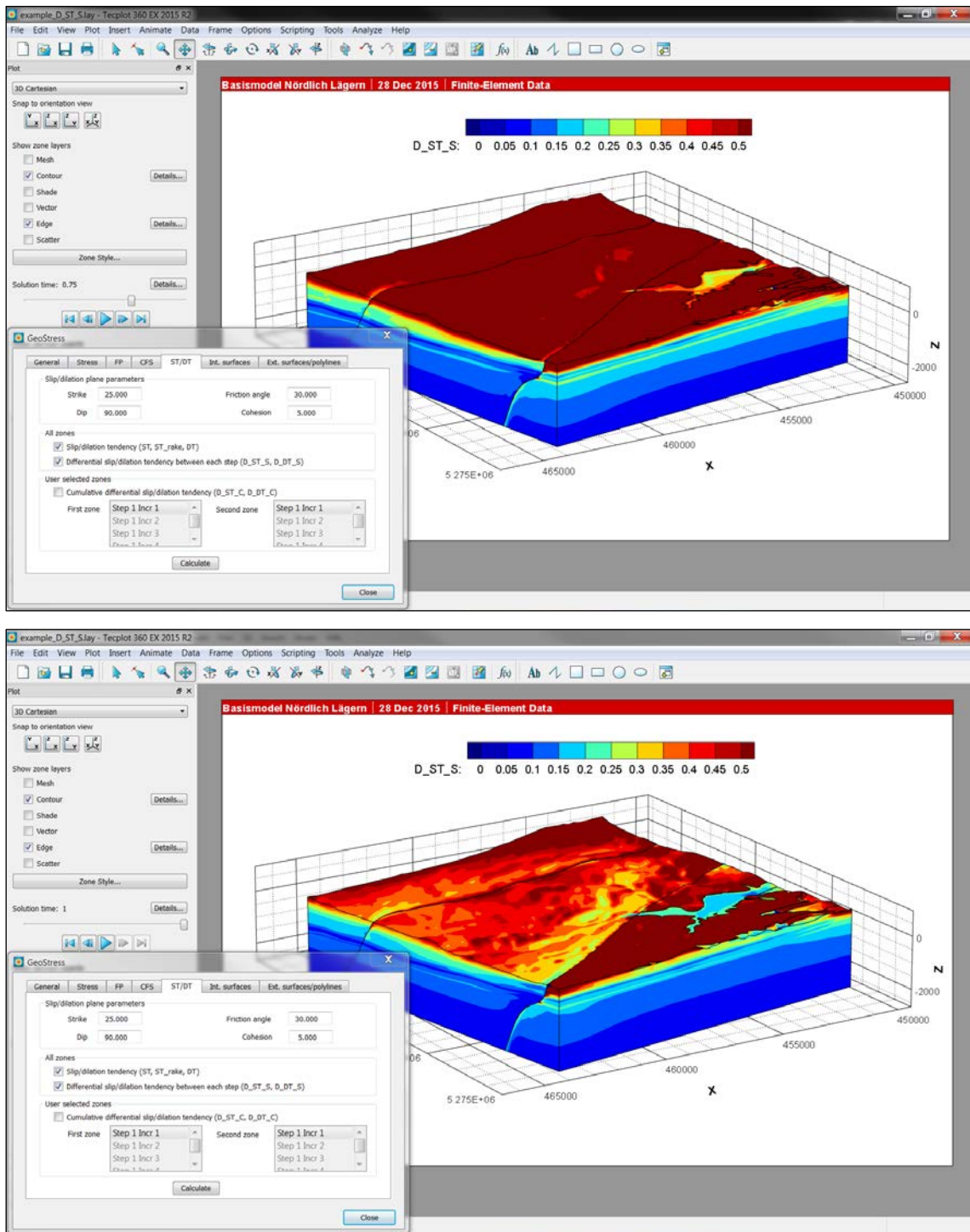


Fig. 3.5-3: Differential slip tendency between all zones.

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.

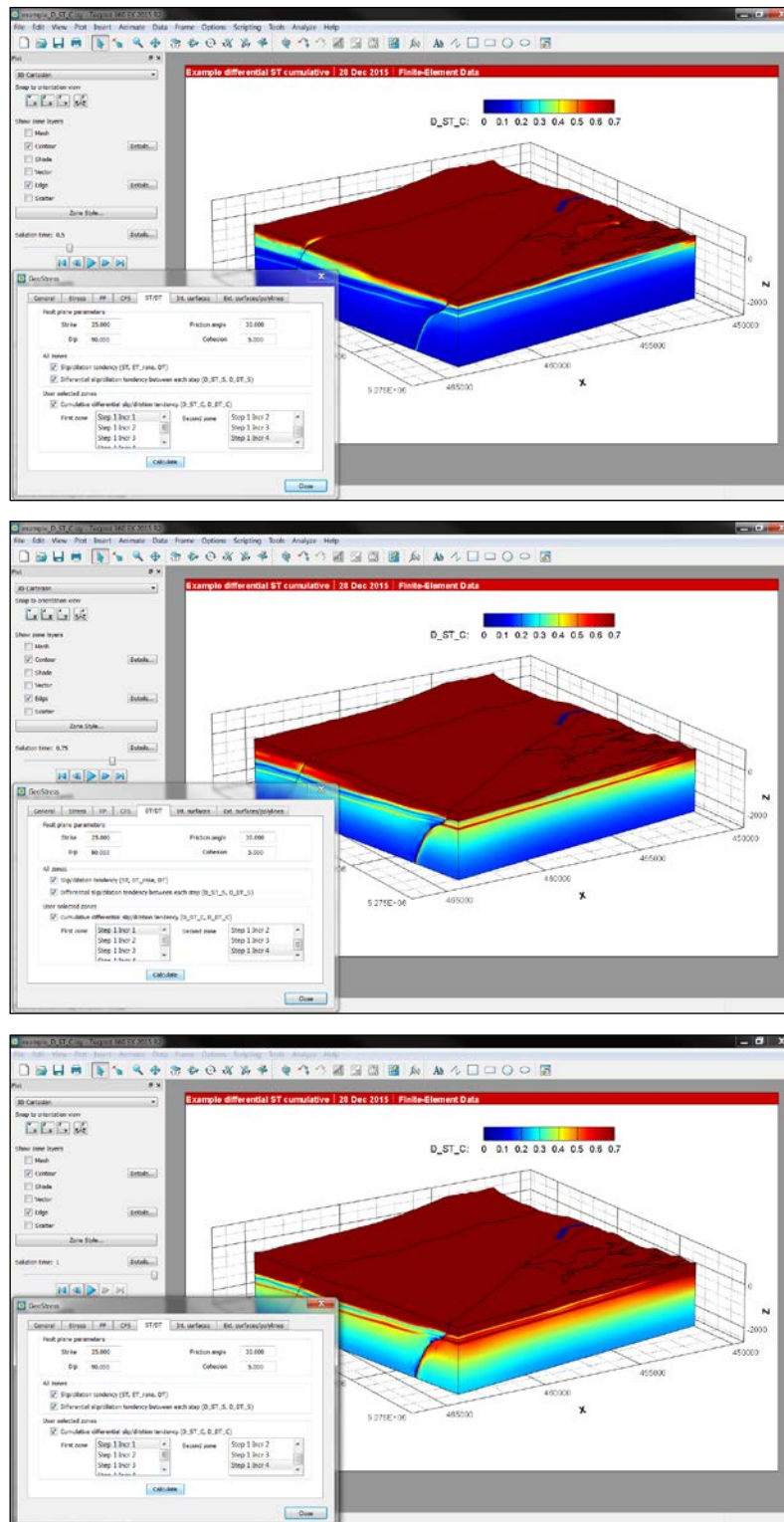


Fig. 3.5-4: Cumulative differential slip tendency between the user defined zones.

Figures show the D_ST_C in MPa for three steps. The upper figure shows D_ST_C for Step1 Incr2 - Step 1 Incr1, the middle one for Step1 Incr3 - Step 1 Incr2 and the lower one for Step1 Incr4 - Step 1 Incr3. The value of D_ST_C that is presented in a zone is always the difference to the selected first zone and cumulative value.

3.6 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 could 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 the external surfaces are generic and they are explicitly not part of the geomechanical-numerical model. They are used to visualize the model results and derived variables at desired 2D regions. This option is described in section 3.7.

With the page *Internal Surfaces* the user can map all variable that exist at the nodes of the finite element mesh including the variable that have been generated with *GeoStress* 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 zones 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. 3.6-1).

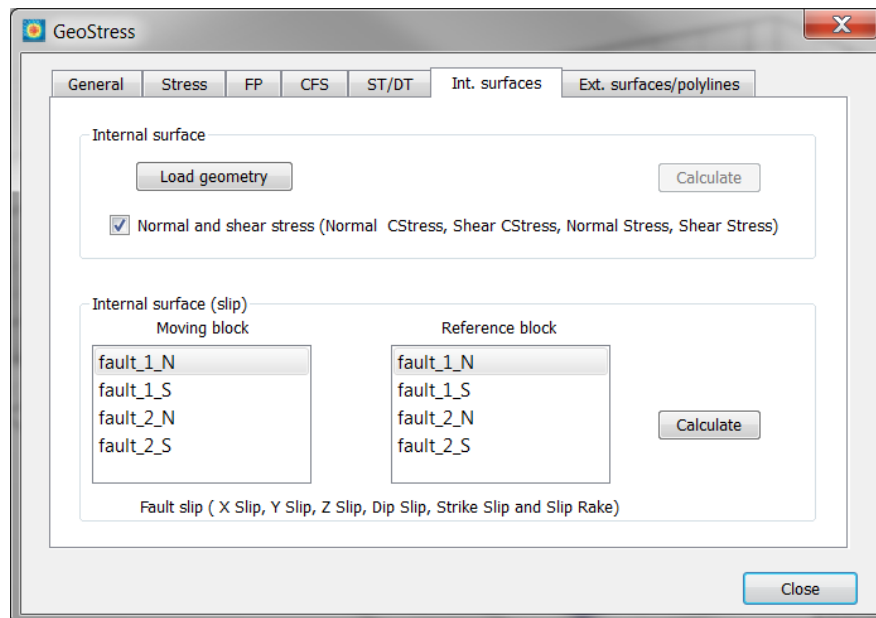


Fig. 3.6-1: Page *Internal Surfaces*.

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

3.6.1 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
      3, 678193.11586966, 275648.08804982, -1166.577839004
      4, 678203.43201998, 275589.58242453, -810.9971133615.
.
.
      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
      828055, 169371, 169229, 169227
.
.
*ELEMENT,TYPE=S4
      762143, 191459, 168028, 168647, 186086
      762204, 366574, 344480, 342914, 367161
      762979, 101237, 101236, 160032, 160035.
.
**

```

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*, *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 tensor *GeoStress* resolves the *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 306 EX is loading these Abaqus variables and renames them to *Contact Pressure* for the normal stress component, and *X Con-*

tact Traction, *Y Contact Traction* and *Z Contact Traction* for the three shear vector components. The *Z Contact Traction* is zero whereas the other two components describe the shear stress on the fault segment. *GeoStress* uses also these values and renames them to the variables *Normal CStress* and *Shear CStress*.

The values from Abaqus and *GeoStress* can deviate from each other since Abaqus is using probably directly the stress tensor values from the integration points or the results from the equilibrium solution procedure for the contact surface. Thus, at surfaces with high curvature or with kinks the deviation can be larger. However, in these cases the artefacts that are generated at these kinks are much larger (and unrealistic) than the deviation between the two approaches. Furthermore, Tecplot 360 EX (version 2015 R2) is not reading the *Contact Pressure* variable correctly. It is only provided for the first contact surface, but not for other contact surface in the model. 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.

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 zones 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...*

3.6.2 Application examples for internal surfaces

In order to generate the results displayed in Figure 3.6-2 (*Normal Stress*) and Figure 3.6-3 (*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. 3.6-2 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 tectonic stresses on the individual segments of the fault.

Fig. 3.6-3 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.

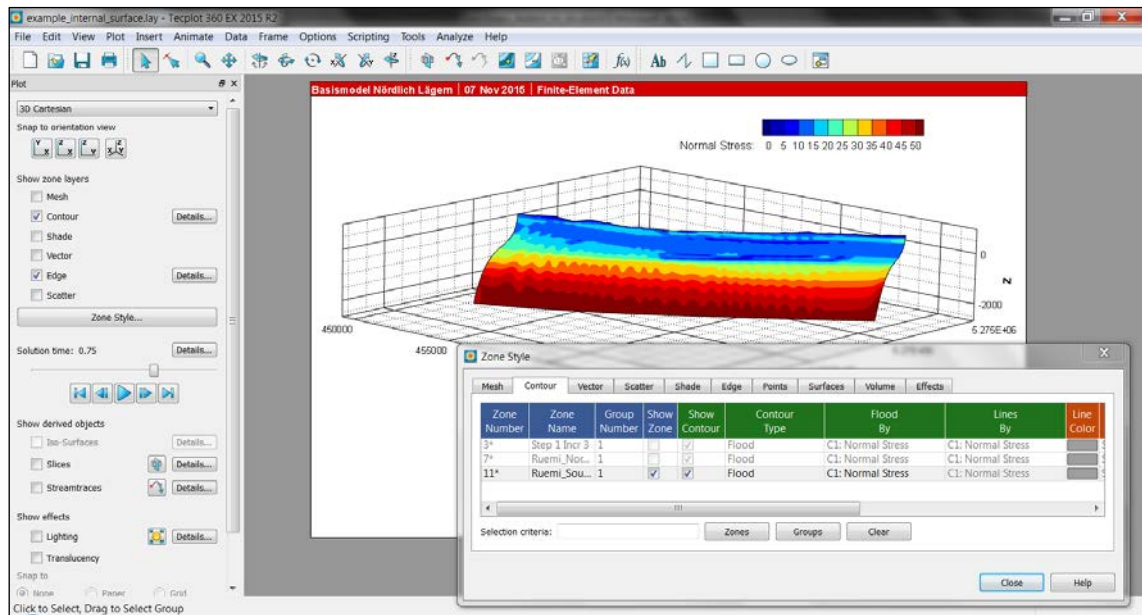


Fig. 3.6-2: Normal stress distribution on an internal surface.

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.

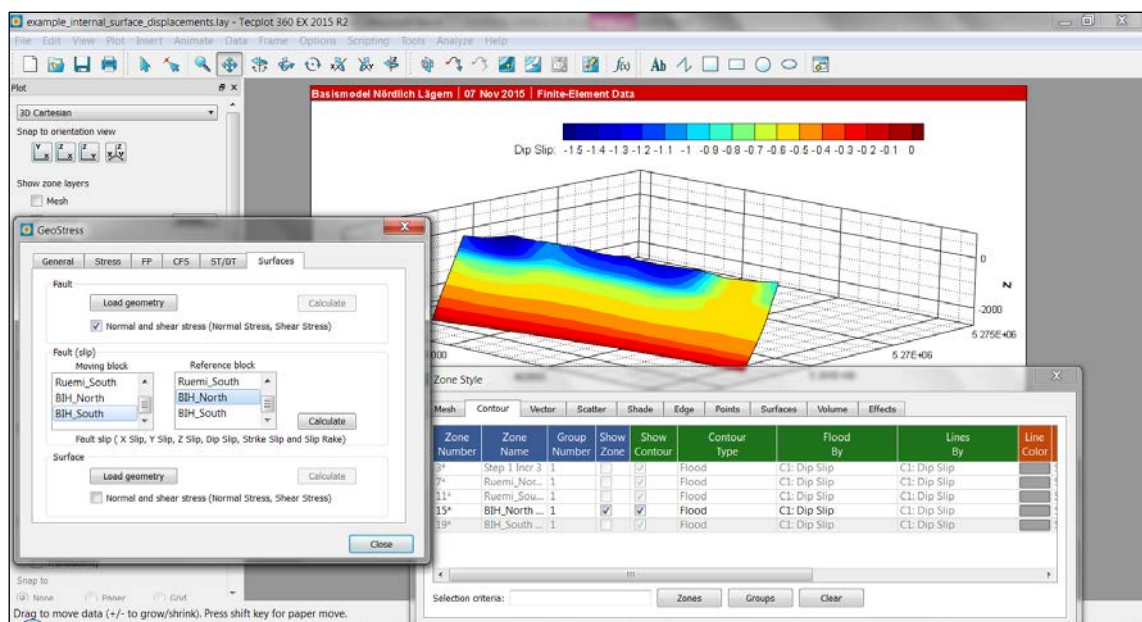


Fig. 3.6-3: Dip slip distribution on an internal surface.

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 zones where internal surfaces are stored. External surfaces cannot have a relative displacement as they are not part of the model.

3.7 External surfaces and polylines

With the page *External Surfaces/Polylines* the user can map all variable that exist at the nodes of the finite element mesh including the variable 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. 3.7-1). In the upper section an external surface can be loaded. In addition to the interpolation of all variable from the model volume the user can 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 given fault plane parameter and the maximum shear stress values. Note that these fault plane parameters are only used for the calculation of the new variable for this fault. The values for slip and dilation tendency as well as for the Coulomb failure stress estimated in the other tab cards of *GeoStress* are not updated. In the lower section a polyline geometry can be imported and all values from the volume are interpolated on this line. For both geometries new zones are generated and can be displayed separately.

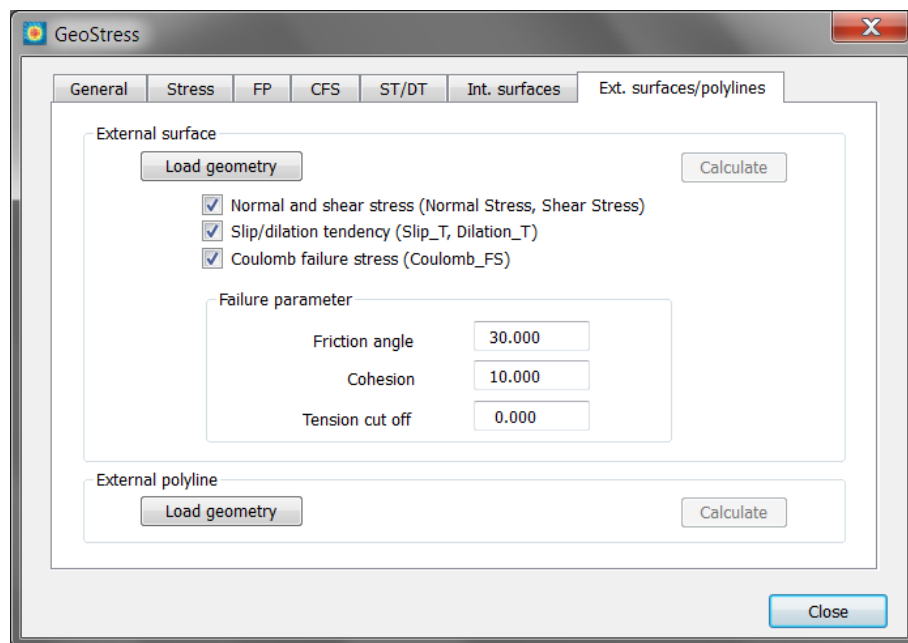


Fig. 3.7-1: Page *External Surfaces/Polylines*.

The page is divided in two sections. In the upper section external surfaces can be loaded in order to interpolate all values on these and to calculate for these normal and shear stresses, slip and dilation tendency (*Slip_T*, *Dilation_T*) as well as Coulomb failure stress (*Coulomb_FS*) using the given failure parameter. Note that these values are different from the ones calculated in the other tab cards. In the lower section an external polyline (e.g. representing a borehole pathway) can be loaded. For both geometries new zones are generated and all stress values are interpolated on these.

The geometry of the surface or polyline has to be imported from a text file with a format that will be described below. This import generates a new set of zones for this geometry only and the results can be displayed over time (increments/steps) on these surfaces without showing the whole volume. I.e. for each increment the map variable can be displayed on the surface.

3.7.1 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
    982774, 688795.9964581, 276139.08833193, -259.2988903596
    982775, 688719.20629053, 276203.53128934, -265.8293048217
    982776, 688640.90023015, 276266.34877857, -271.4864931561
    .
    .
    982777, 688562.55565372, 276328.72343553, -278.5207494841
    985820, 684864.61374959, 277905.62713435, -987.984092963
    985821, 682612.87832328, 277777.36569695, -893.3797783981

*ELEMENT,TYPE=S3
    761142, 985816, 985670, 985821
    761141, 985636, 985821, 985670
    761140, 985430, 985504, 985820
    761139, 985509, 985820, 985504
    .
    .
    755302, 982878, 982879, 982880
    755301, 982868, 982869, 982870
    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*, *SYX*, *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.

3.7.2 External polyline for result visualisation

External polylines are also geometries that were not part of the model. Thus, they are only used in order 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, 458195.00 ,5266455.00, 0.00
2, 458198.78 ,5266457.18, -0.00
3, 458202.56 ,5266459.36, -0.02
899, 460360.05 ,5267704.99, -2491.27
900, 460360.06 ,5267705.00, -2495.64
901, 460360.06 ,5267705.00, -2500.00
**
```

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.

3.7.3 Application example for external surfaces and polylines

In order to generate the results displayed in Figure 3.7-2 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 zones have to be deactivated.

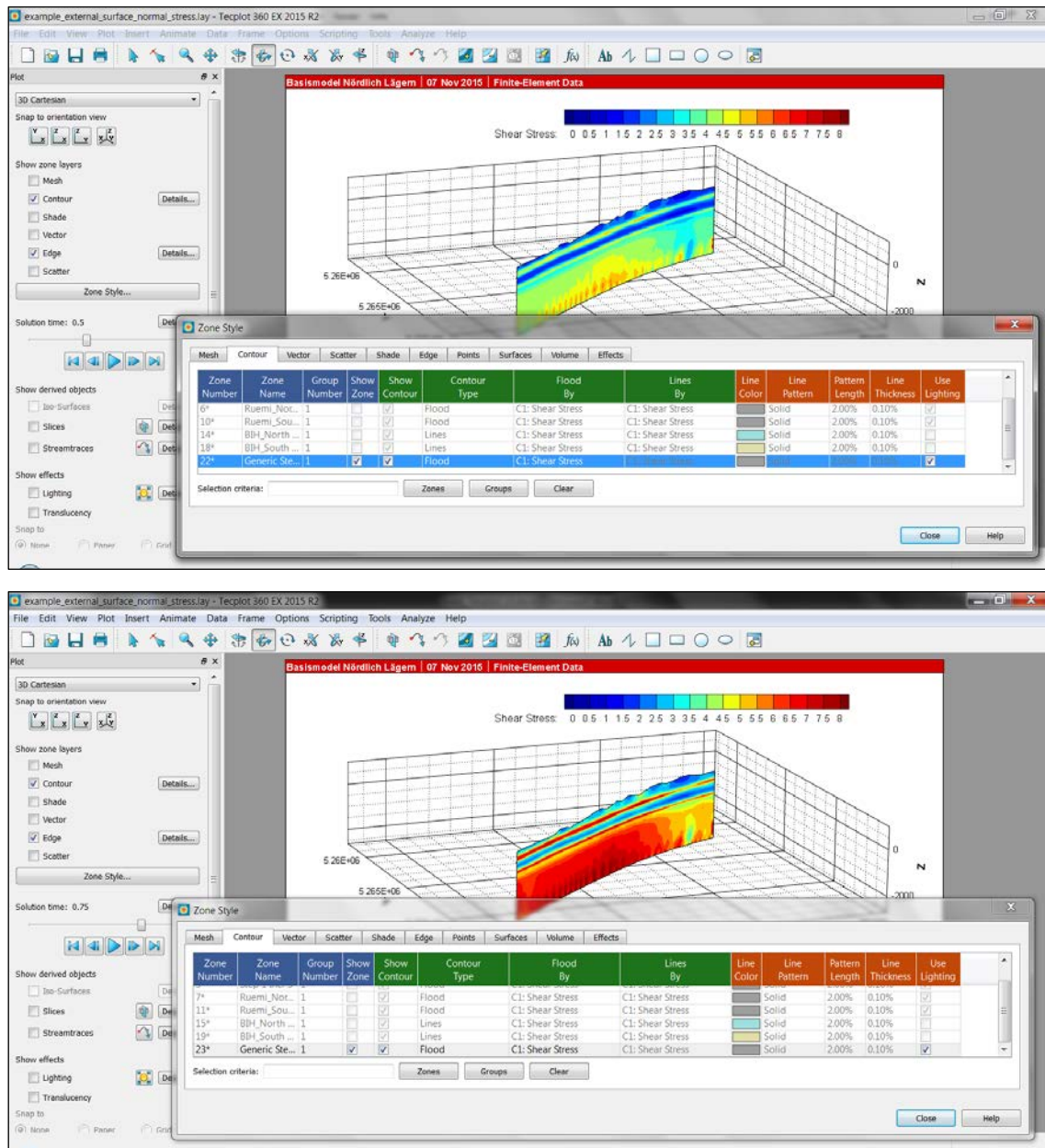


Fig. 3.7-2: Shear stress distribution on an external surface.

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 zones where internal and external surfaces are stored.

3.7.4 Application example for external polylines

In order to generate the results displayed in Figure 3.7-2 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 actually display the values the mesh colour of the polyline has to be changed from a single colour to Multi 1 colour setting (see Fig. 3.7-2). Furthermore, the line thickness of the mesh should be increased to 1.5% and the Edge display option should be turned off.

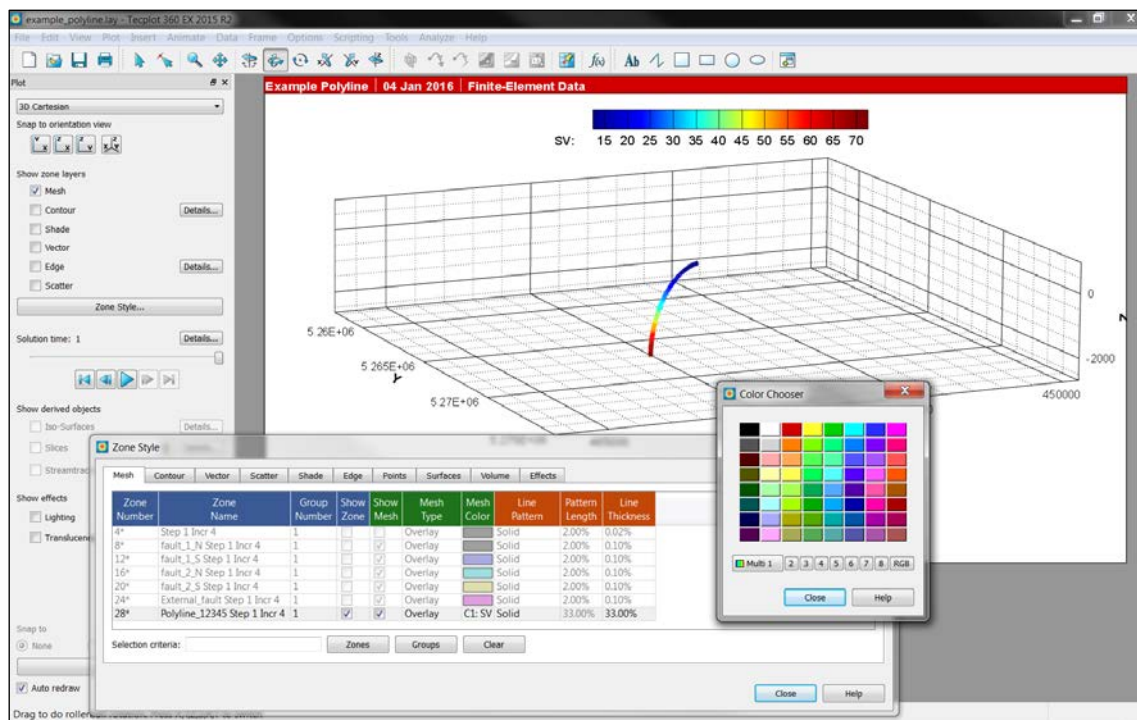


Fig. 3.7-2: Vertical stress along an external polyline.

The vertical stress S_V is calculated for each increment along the polyline. The figure shows the S_V for the fourth solution time 1.0 (Step1 Inc4).

4 General tips for Tecplot 360 EX usage

This chapter presents a number of specific functionalities of Tecplot 360 EX that are probably useful for the analysis of geomechanical-numerical models. Furthermore workflows are presented with examples how to further accelerate the analysis of model results.

4.1 Using Tecplot 360 EX layouts, frame styles and macros

These three options are helpful to generate and load very fast a certain view on your data for a pre-defined variable for specific increment of a specific zone.

Layout files

The layout file 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 zones. The file simply links the view parameter to the *.plt file where the model results are stored in the Tecplot 360 EX format.

Frame styles

A view of a single frame (the active one) can be stored in a file as a so-called frame style in order to apply the chosen settings to another model. It serves as a template to reproduce the view with all details of the setting. An example is given in the folder styles with the name 3D_View_Z_Displacement.sty. However, unfortunately 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*. 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. 4.1-1). When the number of the created variable is not correct anymore, the user can edit the frame style by changing the number.

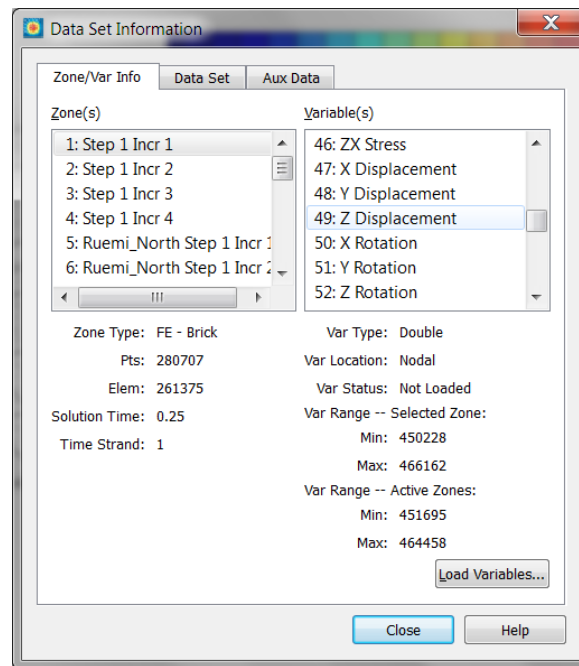


Fig. 4.1-1: Data set information dialog box.

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*).

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 2015
R2'
$!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.

4.2 Write ASCII data from a surface

Tecplot 360 EX allows exporting for any zone or set of zones ASCII data of the model variable results using the menu bar **Tools/Write Data As Formatted Text**. The dialog displayed in Fig. 4.2-1 shows that the user has the full control on what is exported in which format. Of course also the variables calculated with *GeoStress* are accessible.

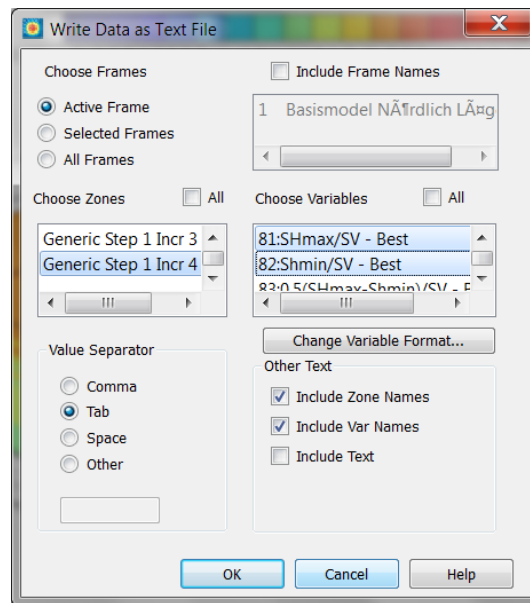


Fig. 4.2-1: Write Data as Text File dialog box.

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

4.3 Calculation of new variables with equations

From the existing variable 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. 4.3-1) 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}'

```

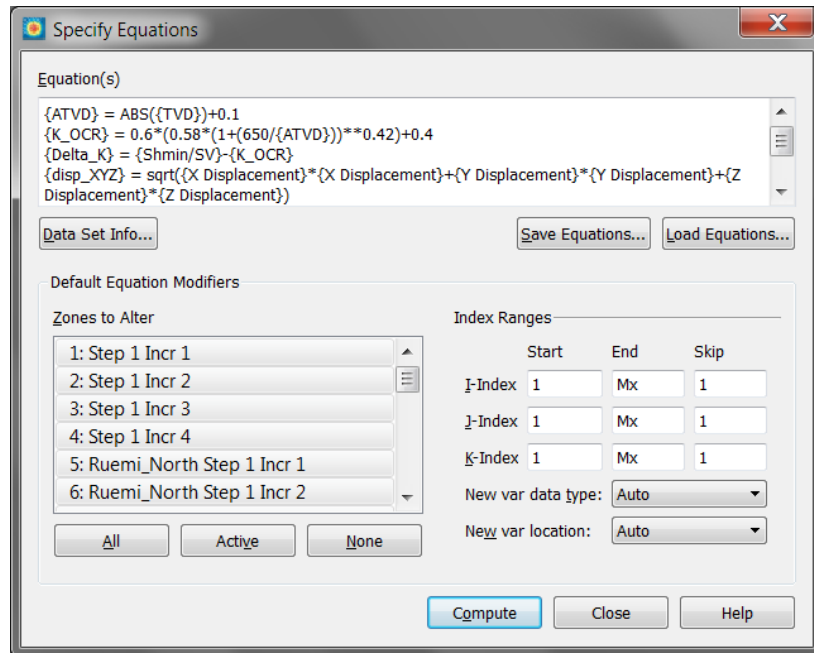


Fig. 4.3-1: Write Data as Text File dialog box.

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

4.4 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. 4.5-1). 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. 4.5-2).

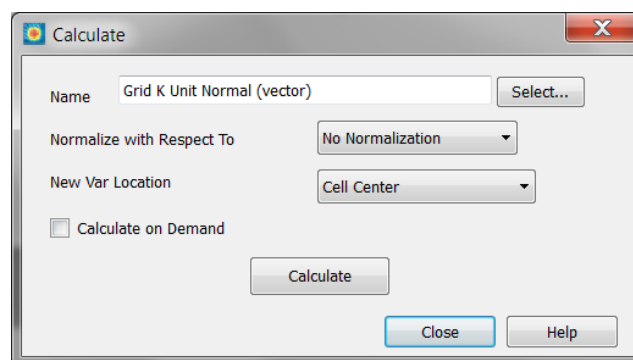


Fig. 4.4-1: Dialog box for calculation of variables.

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

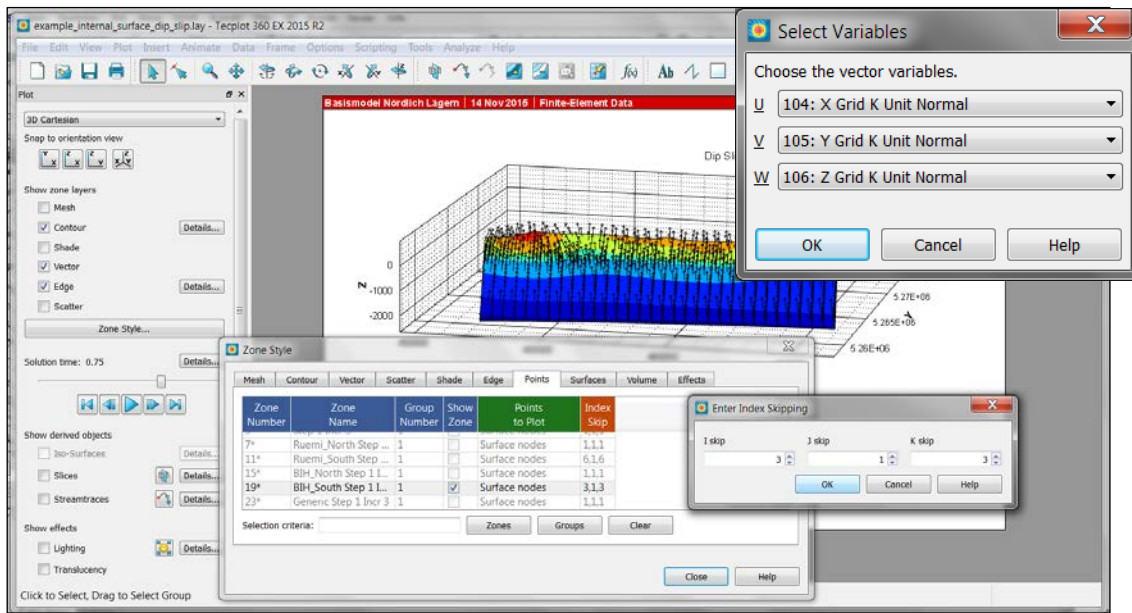


Fig. 4.4-2: Visualization of the normal vector of surfaces.

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 only every third vector should be displayed.

5 References

- Aki, K. and P.G. Richards, 1980, Quantitative Seismology, Vol. 1, W.H. Freeman and Co Ltd.
- Anderson, E.M., 1905, The dynamics of faulting, *Trans. Edin. Geol. Soc.* 8, 387-402.
- Anderson, E.M., 1951, The dynamics of faulting and dyke formation with application to Britian, Oliver and Boyd.
- C  l  rier, B., 1995, Tectonic regime and slip orientation of reactivated faults, *Geophys. J. Int.* 121, 143-161.
- Connolly, P. and J. Cosgrove, 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 & J.J. Wilinkson), pp. 105-121, Geological Society.
- Engelder, T., 1992, Stress Regimes in the Lithosphere.
- Engelder, T., 1994, Deviatoric Stressitis: A Virus Infecting the Earth Science Community, *EOS Trans.* 75, 209, 211-212.
- Friedmann, M., 1972, Residual elastic strain in rocks, *Tectonophys.* 15, 297-330.
- Heidbach, O., M. Tingay, A. Barth, J. Reinecker, D. Kurfe   and B. M  ller, 2010, Global crustal stress pattern based on the World Stress Map database release 2008, *Tectonophys.* 482, 3-15, doi:10.1016/j.tecto.2009.07.023.
- Heidbach, O., T. Hergert, K. Reiter and S. Giger, 2013, Local stress field sensitivity analysis - case study N  rdlich L  gern. In: *NAGRA Arb. Ber. NAB 13-88*, pp. 50.
- Hergert, T., O. Heidbach, K. Reiter, S. Giger and P. Marschall, 2015, Stress field sensitivity analsis in a sedimentary sequence of the Alpine foreland, northern Switzerland, *Solid Earth* 6, 533-552, doi:10.5194/se-6-533-2015.
- Jaeger, J.C., N.G.W. Cook and R.W. Zimmermann, 2007, *Fundamentals of Rock Mechanics*, Blackwell Publishing.
- McGarr, A. and N.C. Gay, 1978, State of Stress in the Earth's Crust, *Ann. Rev. Earth Planet. Sci.* 6, 405-436.
- NAGRA, 2013, Local stress field sensitivity analysis - case study N  rdlich L  gern. In: *NAGRA Arb. Ber. NAB 13-88*, pp. 50.
- Neves, M.C., L.T. Paiva and J. Luis, 2009, Software for slip-tendency analysis in 3D: A plug-in for Coulomb, *Comp. & Geosciences* 35, doi: 10.1016/j.cageo-2009.03.008, 2345-2352.
- Reiter, K. and O. Heidbach, 2014, 3-D geomechanical-numerical model of the contemporary crustal stress state in the Alberta Basin (Canada), *Solid Earth*, doi: 10.5194/se-5-1123-2014.
- Schmitt, D.R., C.A. Currie and L. Zhang, 2012, Crustal stress determination from boreholes and rock cores: Fundamental principles *Tectonophys.*, doi: 10.1016/j.tecto.2012.08.029 1-26.
- Simpson, R.W., 1997, Quantifying Anderson's fault types, *J. Geophys. Res.* 102, 17909-17919.
- Zoback, M., 2010, *Reservoir Geomechanics*, Cambridge.