In-Plane Principal Stress Output in DIANA
Outline

1 Why Displaying In-plane Principal Stresses? 3
2 How Are In-plane Principal Stresses Determined? 4
3 Example 7
4 Finite Element Model 8
   4.1 Geometry ............................................................. 10
   4.2 Properties ........................................................... 13
   4.3 Boundary Conditions ............................................... 14
   4.4 Loads ................................................................. 17
   4.5 Mesh ................................................................. 19
5 Linear Static Analysis 20
   5.1 Commands ........................................................... 20
   5.2 Clipping Planes for Contour Plots ............................... 22
   5.3 Cutting Planes for Tensor Plots ................................. 29
   5.4 Results  .................................................................. 31
      5.4.1 Load Case 1 - Edge Load (2D Problem) .................... 31
      5.4.2 Load Case 2 - Point Load (3D Problem) ................. 36
Appendix A Additional Information 42
1 Why Displaying In-plane Principal Stresses?

A stress rosette is a combination of orthogonal lines in which its lengths correspond to the absolute value of principal stresses and its orientations to the principal directions. The lines are usually colored blue when the principal stress is negative (compression) and red when the principal stress is positive (tension). In the principal coordinate system there are no shear stresses.

Stress rosettes in finite element analysis are used to represent the stress paths in the models. A rosette is represented in the middle of each element. For 2D models these representations are easy to read, because there are only two principal directions presented in planar surfaces. This is not the case for 3D models as the rosettes have 3 orthogonal lines and are located inside the model making cloudy pictures that are very difficult to interpret (example in Figure 1).

![Figure 1: Principal stress rosette output in a 3D model in Fx+ for DIANA](https://dianafea.com)

For this reason, DIANAIE has an alternative approach: principal stresses can be projected to a surface (an outer surface of a cross-section located inside the model). Two-dimensional rosettes with two orthogonal lines are presented in this surface resulting into readable pictures that provide the information about stress orientations in a 3D model. These are the so-called *in-plane principal stresses* represented with 2D rosettes and *normal stress component* displayed as contour plots.

It is a sectional representation of principal stress components that allows to see stress directions at outer and inside surfaces in the 3D model.

This tutorial explains the meaning of in-plane principal stress output in DIANAIE followed by an example.
2 How Are In-plane Principal Stresses Determined?

The in-plane principal stresses are determined in the post-processing module from the stress tensor computed in the DIANA solver. The in-plane principal stresses are computed at the middle of the element using the stress tensor averaged from the stress tensors in the respective nodes.

In a general 3D problem, the principal stresses are the eigenvalues of the stress tensor. The principal directions are the direction of the eigenvalues (eigenvectors). Shear stresses are null in the principal directions.

\[
\sigma = \begin{bmatrix}
\sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\
\sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\
\sigma_{zx} & \sigma_{zy} & \sigma_{zz}
\end{bmatrix}
\]

Eigenvalues = principal stresses
Eigenvectors = principal directions

Figure 2: Global stresses and principal stresses determined in the solver
In plane-principal stress components output in DianaIE are the principal stresses and directions projected in a given surface. These are determined by the following steps:

1. average the 3D stress tensor in the element from the stress tensor in the nodes
2. find the surface desired for the projection
3. rotate the averaged 3D stress tensor such that the z-axis is perpendicular to the surface [Fig. 3]

$$
\sigma = \begin{bmatrix}
\sigma_{xx} \\
\sigma_{yy} \\
\sigma_{zz} \\
\sigma_{xy} \\
\sigma_{yz} \\
\sigma_{zx}
\end{bmatrix}
$$

Referential transformation

$$
\sigma' = Q \sigma Q^T
$$

Q is the transformation matrix with the director cosines

Figure 3: Surface for projection of principal stresses and referential transformation
4. from the in-plane components \([\sigma_{xx}, \sigma_{yy}, \sigma_{xy}]\) compute the 2D principal stresses and directions \([\sigma_1, \sigma_2]\) that are displayed as a rosette [Fig. 4]

5. the normal stress component \(\sigma_{zz}\) is displayed as a contour plot in the surface

\[
\begin{bmatrix}
\sigma'_{xx} \\
\sigma'_{yy} \\
\sigma'_{xy}
\end{bmatrix}
\]

\[
\sigma'_{IP} = \begin{bmatrix}
\sigma'_{1} \\
\sigma'_{2} \\
\sigma'_{xy}
\end{bmatrix}
\]

Eigenvalues = principal stresses
Eigenvectors = principal directions

\[
\sigma'_{zz}
\]

Figure 4: Determination of in-plane principals and normal stress component in the output
3 Example

In the following example we illustrate the use of in-plane principal stress output in DIANAIE.

The model is a box with dimensions of height = 2 m, width = 1 m and length = 10 m, supported at two end bottom edges (at begin length and at end length). The material is linear elastic with Young’s modulus of 30E+7 N/m² and null Poisson ratio.

We study two load cases:

1. edge load applied at top surface, middle length in the height direction (representing a pure 2D configuration modeled with 3D mesh) [Fig. 5]
2. point load applied at top surface, middle length, begin width in height direction (generating torsion deformations in the model and making it a real 3D problem) [Fig. 6].

We perform a linear elastic analysis. We want to see the global and principal stress tensors in the box under the two load cases.

In the load case 1 (2D problem) in-plane principal stress components directly correspond to the solver principal stresses as there are no shear components.

In the load case 2 (3D problem) the in-plane principal stress components differ from the solver principal stresses as the shear components are not zero.

Figure 5: Box (2 × 1 × 10 m) under edge load case

Figure 6: Box (2 × 1 × 10 m) under point load case
4 Finite Element Model

For the modeling session we start a new project. We make a 3D structural model and use linear hexagonal elements.

Main menu → File → New  [Fig. 7]

Figure 7: New project dialog
We use meter for the unit length and newton for force (SI units).
4.1 Geometry

We start the model by making a polygon line in \( Y = 0 \) and \( Z = 0 \) with the length of the block (10 m) and including a vertex in the middle of the line. We extrude this line 1 m in the \( Y \) direction (width) [Fig. 11] and 2 m in the \( Z \) direction (height) [Fig. 12]. We rename the shape ‘Polyline 1’ to ‘Block’. In this manner we have the block [Fig. 13] with a line in the middle of the top surface needed to define the edge load. With this procedure we also ensure an uniform mesh.

---

**Main menu** → **Geometry** → **Create** → **Add polyline** [Fig. 10]

**Main menu** → **Geometry** → **Modify** → **Extrude** [Fig. 11] [Fig. 12]

**Geometry browser** → **Geometry** → **Shapes** → **Polyline 1** → **Rename** → **Block**
We create a vertex in the middle of the bottom edge in order to later define a support condition there. To be part of the geometry we need to imprint the vertex in the block.
4.2 Properties

We define a linear elastic material with Young’s modulus of $3.0\times10^7$ N/m$^2$ and null Poisson ratio. We use solid elements and we don’t need to define geometry properties. We don’t define a data set because we use the default integration scheme.

---

**Main menu** → Geometry → Assign → Shape Properties [Fig. 18]

Shape Properties [Fig. 19] → Material → Add material Edit material [Fig. 20]

---

![Figure 18: Assign properties](image1)

![Figure 19: Add new material - linear elastic](image2)

![Figure 20: Material properties](image3)
4.3 Boundary Conditions

We restrain the translations in the $Z$ direction in both extreme edges along the width at the bottom of the block.

Main menu ➔ Geometry ➔ Assign ➔ Add supports  

[Fig. 21]

Figure 21: Add edge support

Figure 22: View of the model - edge support
We restrain the translations in the X direction in one edge along the width at the bottom of the block.

**Main menu** ➔ Geometry ➔ Assign ➔ Add supports ➔ [Fig. 23]

**Figure 23:** Add edge support

**Figure 24:** View of the model - edge support
To prevent rigid body displacements, we restrain the translation in the $Y$ direction in the vertex defined in the bottom edge of the block.

Main menu ➔ Geometry ➔ Assign ➔ Add supports ➔ [Fig. 25]

Figure 25: Add point support

Figure 26: View of the model - point support
4.4 Loads

We start defining the first load case - *Edge load* - a vertical line load of -10 N/m applied at the top surface at middle length.

**Main menu** ➔ Geometry ➔ Assign ➔ Add loads 📦 [Fig. 27]

**Figure 27:** Edge load

**Figure 28: Model view - edge load**
We define the second load case - *Point load* - a vertical point load of -10 N applied at the top surface at middle length, begin width.

**Main menu ➔ Geometry ➔ Assign ➔ Add loads 🏷️ [Fig. 29]**

**Figure 29: Point load**

**Figure 30: Model view - point load**
4.5 Mesh

We set the element size as 0.15 m and generate the mesh.
5 Linear Static Analysis

5.1 Commands

We set a structural linear analysis with the default characteristics.

Main menu  ➔  Analysis  ➔  Add analysis  📝 [Fig. 34]
Analysis browser  ➔  Analysis1  ⚪  ➔  Add command  ➔  Structural linear static  [Fig. 35] [Fig. 36]
We specify the desired output results: stresses in global and principal directions and displacements [Fig. 38]. As in-plane principal stress components are calculated from the average element stresses, we ask also for this specific output in order to allow for a direct comparison [Fig. 39]. Then, we run the analysis.

**Analysis browser** → **Analysis1** → **Structural linear static** → **Output linear static analysis** → **Edit properties** [Fig. 37]

Properties - **OUTPUT** → Result → **User selection** → **Modify** [Fig. 38]

< Repeat for **STRESS TOTAL CAUCHY PRINCIPAL** >

**Main menu** → **Analysis** → **Run selected analysis**

Figure 37: Edit output properties

Figure 38: Selection of results

Figure 39: Result properties

Figure 40: Edit output properties
5.2 Clipping Planes for Contour Plots

We use clipping planes to represent contour plots of stresses in different surfaces of the model. We want to study the stress state in four surfaces:

1. front face $Y = 0$ (length $\times$ height);
2. back face $Y = 1$ (length $\times$ height);
3. section $X = 5$ (height $\times$ width);
4. section $X = 6$ (height $\times$ width).

Figure 41: Surfaces for stress output representation
So, first we need to define the four clipping panes for these surfaces. We start by setting four clipping panes.

![Main menu](https://dianafea.com)

**Property Panel**
- Show view settings
- Result
- Contour plot settings
- Clip Settings
- Add...
- Plane

*Fig. 42* [Fig. 43]  
< Rename the four planes as $Y=0$, $Y=1$, $X=5$, $X=6$ (right click the names) [Fig. 44]  >

<table>
<thead>
<tr>
<th>Description</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Common</td>
<td></td>
</tr>
<tr>
<td>Geometry</td>
<td></td>
</tr>
<tr>
<td>Mesh</td>
<td></td>
</tr>
<tr>
<td>Result</td>
<td></td>
</tr>
<tr>
<td>Contour plot settings</td>
<td></td>
</tr>
<tr>
<td>Equidistant levels</td>
<td></td>
</tr>
<tr>
<td>Equidistant levels options</td>
<td></td>
</tr>
<tr>
<td>Plot iso-surfaces at levels</td>
<td></td>
</tr>
<tr>
<td>Color scale type</td>
<td></td>
</tr>
<tr>
<td>Discrete color s...</td>
<td></td>
</tr>
<tr>
<td>Color scale limits</td>
<td></td>
</tr>
<tr>
<td>Auto-scale to di...</td>
<td></td>
</tr>
<tr>
<td>Bounding colors</td>
<td></td>
</tr>
<tr>
<td>Clip settings</td>
<td></td>
</tr>
<tr>
<td>Enable</td>
<td></td>
</tr>
<tr>
<td>Add...</td>
<td></td>
</tr>
<tr>
<td>Slice</td>
<td></td>
</tr>
<tr>
<td>Show cut edge</td>
<td></td>
</tr>
<tr>
<td>Boundary edge color</td>
<td>#a999a9</td>
</tr>
<tr>
<td>Display contour edges</td>
<td></td>
</tr>
<tr>
<td>Vertices as spheres</td>
<td></td>
</tr>
<tr>
<td>Vortex size</td>
<td>6</td>
</tr>
<tr>
<td>Lines as tubes</td>
<td>4</td>
</tr>
<tr>
<td>Line size</td>
<td>4</td>
</tr>
<tr>
<td>Background shading</td>
<td></td>
</tr>
<tr>
<td>Diagram plot settings</td>
<td></td>
</tr>
<tr>
<td>Vector plot settings</td>
<td></td>
</tr>
<tr>
<td>Tensor plot settings</td>
<td></td>
</tr>
<tr>
<td>Crack plot settings</td>
<td></td>
</tr>
</tbody>
</table>

**Figure 42: Add clipping plane**  
**Figure 43: Clipping planes**  
**Figure 44: Rename clipping planes**
We now define the position of each clipping pane.

**Figure 45:** Clip plane settings $Y=0$

**Figure 46:** Clip plane settings $Y=1$
Note: for the outer surfaces we don’t exactly set \( Y = 0 \) and \( Y = 1 \) but an approximate number to place the surfaces inside the model (\( Y = 0.0001 \) and \( Y = 0.9999 \), respectively). For the \( X = 5 \) surface we also give an approximate value (\( X = 5.0001 \)) in order move away from the symmetry point. For the outer surfaces we could see the results without the clipping planes. However, the clipping panes we can do the contour plots only for these surfaces and not for the entire mesh which make it easier to analyze the results.
The *Enable plane* option allows to activate and deactivate the view of each clipping plane. This is useful if we want to show only one plane at a time. In the clip settings we enable the slice option to represent only the desired surface. For now we keep all clipping planes as enable (default setting). We exemplify this with contour plots for stresses SXX [Fig. 51] where we see the results in all the clipping planes. In [Fig. 50] we can also find the global and principal components of the stress tensor.
If we enable just one clipping plane at a time we can see the results in each surface individually.

**Property Panel** ➔ **Result** ➔ **Contour plot settings** ➔ **Clip Settings** ➔ **Y=0** ➔ **Enable plane**  
**Property Panel** ➔ **Result** ➔ **Contour plot settings** ➔ **Clip Settings** ➔ **Y=1** ➔ **Enable plane**  
**Property Panel** ➔ **Result** ➔ **Contour plot settings** ➔ **Clip Settings** ➔ **X=5** ➔ **Enable plane**  
**Property Panel** ➔ **Result** ➔ **Contour plot settings** ➔ **Clip Settings** ➔ **X=6** ➔ **Enable plane**

**Fig. 52**: SXX in plane Y=0  
**Fig. 53**: SXX in plane Y=1  
**Fig. 54**: SXX in plane X=5  
**Fig. 55**: SXX in plane X=6
Tip: it is possible to edit the clipping planes in the *Graphics Window*, by moving the plane with the mouse and place it in any location and direction inside the model. We exemplify this by editing the plane $Y=0$.

**Graphics window** $\rightarrow$ Clipping planes $\rightarrow$ Edit $Y=0$  [Fig. 56]

*Move the position of the clipping plane with the mouse*  [Fig. 57]
5.3 Cutting Planes for Tensor Plots

For the representation of the in-plane principal stresses we need to define cutting planes. In this case, we can only define a plane at a time, so we need to change the location for each surface. The location and normal definitions are the same as defined previously for the clipping planes.

---

**Property Panel**
- Result → Tensor plot settings → Cutting plane settings → Enable

**Figure 58**: Cutting plane settings

**Figure 59**: Cutting plane settings $Y=0$

**Figure 60**: Cutting plane settings $Y=1$

**Figure 61**: Cutting plane settings $X=5$

**Figure 62**: Cutting plane settings $X=6$

---

In-Plane Principal Stress Output in DIANA | https://dianafea.com
We show the location of the different cutting plates with the representation of in-plane principal components (here we can also find the normal principal component output [Fig. 63]).

Results browser → Analysis1 → Output linear static analysis → Element results → Cauchy Total Stresses → SXX → Show in-plane principal components [Fig. 63]

< Change the location of the different cutting planes as in [Fig. 64] - [Fig. 67] >
5.4 Results

We use the clipping and cutting planes defined previously to analyze the different stress components acting in the block. The steps for visualization in DIANAIE are not repeated here. We focus this part of the tutorial on the discussion of the in-plane principals output.

5.4.1 Load Case 1 - Edge Load (2D Problem)

Figure 68 presents the total stresses SXX in the different surfaces (see Figure 50). In the following we see the global and principal stresses and rosettes for each surface in detail. For that we activate only one clipping plane at a time (see Figure 59 to Figure 62 and use left view of the model).

Figure 68: Total stress contours displayed in the clipping planes
We present the contour plots of the global stress tensor $S_{XX}$ and $S_{ZZ}$ and of the principal stresses $S_1$ and $S_3$ in the front face (plane $Y=0$).

In Figure 69 we see the $S_{XX}$ compression (negative values in blue) and tension (positive values in red) zones in the central part, which correspond to the principal stresses [Fig. 71] [Fig. 72] (as these are oriented in length direction).

Towards begin and end parts of the block the principal stresses rotate to the supports. $S_{XX}$ values are not exactly equal to $S_1$ and $S_3$ because there is a concentration of $S_{ZZ}$ [Fig. 70] in the supports and loading points. Extreme values of in-plane principals are the maximum $S_{XX}$ and minimum $S_{ZZ}$.

Note: as the *Edge Load* is symmetric, plane $Y=1$ (back face) presents the same results as plane $Y=0$ (front face) and is not presented for this case.
We now present the in-plane principal components as rosette and the normal principal component as contour plot (see Figure 63).

By comparing Figure 73 with Figure 71 and Figure 72 we see that the in-plane principals correspond to the principal stresses from the solver.

In Figure 74 we see that the normal component is zero in the areas apart from the load concentration areas; in these areas it correspond to stress SZZ (compare with Figure 70).

Figure 73: In-plane principal stresses in cutting plane Y=0

Figure 74: Normal stress component in cutting plane Y=0
We display the stress components (global and principal) in the cross-section correspondent to plane $X=5$.

As this is a load concentration area the in-plane principal stress components are not zero [Fig. 79]. The in-principal stresses displayed as rosettes are approximately null in the bottom (where we have mainly SXX stresses [Fig. 75]) and approximately equal to SZZ [Fig. 76] in the top of the cross section. In the center length we have high SZZ stresses, so we have not null in plane stress components. The normal component show the compressive and tension zones [Fig. 80].

Figure 75: SXX in clipping plane $X=5$

Figure 76: SZZ in clipping plane $X=5$

Figure 77: S1 in clipping plane $X=5$

Figure 78: S3 in clipping plane $X=5$

Figure 79: In-plane principal stresses in cutting plane $X=5$

Figure 80: Normal stress component in cutting plane $X=5$
We now display the same stress components (global and principal) in the cross-section correspondent to plane X=6 (out of the load concentration zone).

Here the in-plane principals [Fig. 85] are approximately zero because the stresses appear predominantly in XX direction. So, the normal component [Fig. 86] is similar to the global stress SXX [Fig. 81], showing the compressive and tension zones.

Figure 81: SXX in clipping plane X=6
Figure 82: SZZ in clipping plane X=6
Figure 83: S1 in clipping plane X=6
Figure 84: S3 in clipping plane X=6
Figure 85: In-plane principal stresses in cutting plane X=6
Figure 86: Normal stress component in cutting plane X=6
5.4.2 Load Case 2 - Point Load (3D Problem)

Figure 87 presents the Cauchy Total Stresses $S_{XX}$ in the different surfaces. In the following we see the global and principal stresses and rosettes for each surface in detail. For that, we define one clipping plane at a time (see Figure 59 to Figure 62) and use left view of the model.

![Figure 87: Contour clips](https://dianafea.com)
We present the contour plots of the global and principal stresses (see Figure 50) in the front face (plane $Y=0$) (see Figure 59).

- Figure 88: $S_{XX}$ in clipping plane $Y=0$
- Figure 89: $S_1$ in clipping plane $Y=0$
- Figure 90: $S_{YY}$ in clipping plane $Y=0$
- Figure 91: $S_2$ in clipping plane $Y=0$
- Figure 92: $S_{ZZ}$ in clipping plane $Y=0$
- Figure 93: $S_3$ in clipping plane $Y=0$

In-Plane Principal Stress Output in DIANA | https://dianafea.com
We also present the in-plane principal components and the normal principal component.

Figure 94: In-plane principal stresses in cutting plane Y=0

Figure 95: Normal stress component in cutting plane Y=0
Now the point load generates a 3D state of stresses with torsion and the front and back faces do not have the same results as in the case of the edge load. So we also present the contour plots of the global and principal stresses in the back face (plane Y=1) (see Figure 60).

Figure 96: SXX in clipping plane Y=1
Figure 97: S1 in clipping plane Y=1
Figure 98: SYY in clipping plane Y=1
Figure 99: S2 in clipping plane Y=1
Figure 100: SZZ in clipping plane Y=1
Figure 101: S3 in clipping plane Y=1
We present the in-plane principal components and the normal principal component for this surface. The in-plane and normal principal components at both outer faces (Y=0 and Y=1) present different results as consequence of eccentricity of loading (compare Figure 94 with Figure 102 for in-plane components and Figure 95 with Figure 103 for normal component).

**Figure 102:** In-plane principal stresses in cutting plane Y=1

**Figure 103:** Normal stress component in cutting plane Y=1
We display the stress components (global and principal) in the cross-section correspondent to plane X=6 (out of the load concentration zone). In the rosette representation (in-plane principals) [Fig. 110] we can observe the torsion effects in the cross section, with the circular flow of shear stresses. The out of plane component [Fig. 111] shows the tensile and compression areas that are not always symmetric and aligned anymore. We can also see the principal values in the cross section that present a non-symmetric pattern [Fig. 107 to 109] (compare these with [Fig. 83 to 84]).
Appendix A  Additional Information

Folder: Tutorials/InPlanePrincipals

Number of elements ≈ 5200

Keywords:
- ANALYS: linear static.
- CLASS: large.
- CONSTR: support.
- ELEMEN: hx24l solid.
- LOAD: edge elemen force node.
- MATERI: elasti isotro.
- OPTION: direct.
- POST: binary ndiana.
- PRE: diana.
- RESULT: cauchy displa princ stress total.
Disclaimer: The aim of this technical tutorial is to illustrate various tools, modelling techniques and analysis workflows in DIANA. DIANA FEA BV does not accept any responsibility regarding the presented cases, used parameters, and presented results.