European Commission

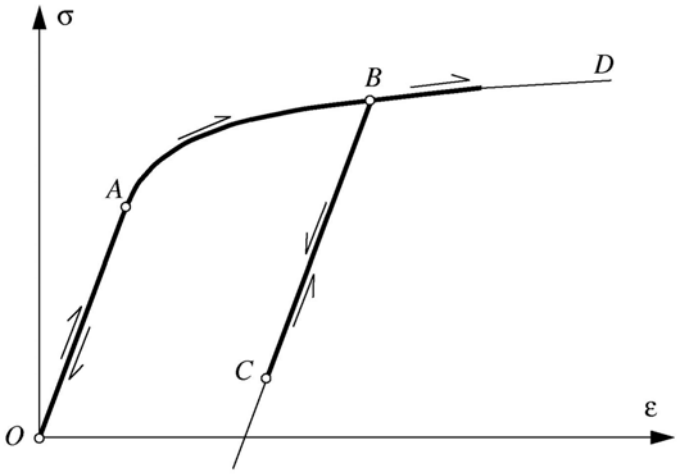
J R C T E C H N I C A L R E P O R T S

Some notes on the organization of routines for solid materials in EUROPLEXUS

Folco Casadei
Georgios Valsamos
Martin Larcher
Vegard Aune

2015

Report EUR 27683 EN



epx europlexus software

Joint Research Centre

European Commission
Joint Research Centre
Institute for the Protection and Security of the Citizen

Contact information
Georgios Valsamos
Address: Joint Research Centre, Via Enrico Fermi 2749, TP 480, 21027 Ispra (VA), Italy
E-mail: georgios.valsamos@jrc.ec.europa.eu
Tel.: +39 0332 78 9004
Fax: +39 0332 78 9049

http://ipsc.jrc.ec.europa.eu/
http://www.jrc.ec.europa.eu/

This publication is a Technical Report by the Joint Research Centre of the European Commission.

JRC97565

EUR 27683 EN

ISBN 978-92-79-54259-6 (PDF)

ISSN 1831-9424 (online)

doi:10.2788/98573

Luxembourg: Publications Office of the European Union, 2015

Printed in Italy

Technical Note

Some notes on the organization of routines for solid materials in EUROPLEXUS

F. Casadei, G. Valsamos, M. Larcher

European Commission, Institute for the Protection and Security of the Citizen
Joint Research Centre, 21027 Ispra, Italy

V. Aune

Structural Impact Laboratory (SIMLab), Department of Structural Engineering,
Norwegian University of Science and Technology,
Rich. Birkelands vei 1A, NO-7491 Trondheim, Norway

Contents

1 Introduction

This document describes the organization of some routines for the modelling of solid materials in the EUROPLEXUS computer code (EPX) at the Joint Research Center (JRC) of the European Commission (EC) at Ispra.

EPX is a computer code jointly developed by the French Commissariat à l'Energie Atomique (CEA DMT Saclay) and by EC-JRC IPSC. The code application domain is the numerical simulation of fast transient phenomena such as explosions, crashes and impacts in complex three-dimensional fluid-structure systems.

The EPX code contains libraries of element types (about 150 types) and of material types (about 120 types). See [1] for details.

Elements cover 1D, 2D and 3D geometrical situations. Some of them (namely some general-purpose continuum elements) can be used to discretize both the structural and the fluid domain. Others are specific to either fluid or structural applications. To the latter category belong the so-called "structural" elements, namely bars, beams and shells.

Materials are either fluid or solid (structural). Only some (meaningful) combinations of elements and materials are allowed in the code. For example, a structural element can be assigned a solid material, but not a fluid material.

This document illustrates some practical and programming aspects on the organization of solid material models in EPX in order to provide a reference for adding new material models of this type in the code.

2 Code organization

The organization of the code as concerns the implementation of material models for solids is now presented in some detail.

2.1 General code organization

The general organization of the code can be schematized as follows (by greatly simplifying):

- Loop on time steps
    - Loop on elements
        - Call element routine
            - Element routine calls the material routine for the current element
    - End of loop on elements
- End of loop on time steps

EPX is an explicit code for transient dynamics and the typical formulation of constitutive laws uses rate (or incremental) form. The element routine typically performs geometrical calculations in order to compute strain increments. Then it calls the appropriate material routine (among the various possible materials which are available for the current element type) in order to update the stress tensor. Finally, it integrates the stresses over the element in order to obtain the vector of nodal internal forces.

In this type of constitutive laws, the material routine typically receives in input (at least):

- the set of (constant) parameters for the present material. These depend only on the material (not on the Gauss point) and are never modified by the material routine;
- the (old) stress state, the (old) set of internal variables, and the strain increments at the current Gauss point

and uses these to update both the stresses and the internal variables at the current Gauss point.

Typically, structural elements possess several "Gauss" points (or similar integration points) used for numerical integration of stresses in order to obtain the nodal internal forces). In such a case, a loop over such points is performed in the element routine and the appropriate material routine is called once for each Gauss point concerned.

An element can be homogeneous, i.e. it possesses only one material, or heterogeneous. The latter case applies for example to layered shell elements, for which each layer may possess a different material. In such a case, for the same element each Gauss point may call a different material routine.

2.2 Constitutive model routines

In EPX each material model is programmed in a specific Fortran 90 (F90) module. The material name consists of (up to) 4 letters, e.g. LINE for a linear elastic material. The corresponding module is named M_MATERIAL_xxxx.FF (M_MATERIAL_LINE.FF in the example), where .FF is the standard extension of EPX source files.

Note that these files are not compiled directly but are first transformed into compilable F90 files (with the extension .F) by "filtering" them with a suitable tool. The filtering mechanism is mainly used to deal with platform dependencies. The .FF version of the file can contain code suitable for all the various platforms on which EPX can run, while the .F version will contain only the code for the chosen platform. Normally in constitutive model routines there are no platform dependencies, so that the .FF and .F versions of the source are typically identical.

The typical material module contains the following routines:

- A routine that reads the material constants from the input data set. For example, a linear elastic material would need (at least) Young's modulus E and Poisson's coefficient ν, in addition to the material density ρ which, however, is typically not used in the constitutive law.

- A routine that implements the constitutive law for the material (constitutive routine). As anticipated in Section 2.1, the scope of this routine is to increment the stresses and the material's internal variables in a generic Gauss point. The internal variables can be hardening parameters, stress invariants, accessory variables, etc., and in general any property which varies both in time (as the deformation process goes on) and in space (from Gauss point to Gauss point).

- Any number (including zero) of accessory routines needed by the present material model and called by the previous routine. These are typically private routines to the module, i.e. their names are not exported and are not visible from other parts of the code.

Since EPX has many element types, each material (especially a solid material) can be used in a variety of geometrical and loading situations. For example, a material for ductile plastic behaviour typical of metals can be associated with a bar (in 2D or 3D), a 2D beam or shell, a 2D continuum element, a 3D beam or shell, or a 3D continuum element. Furthermore, the 2D continuum case can use either plane stress, or plane strain, or axisymmetric formulation.

A separate version of the constitutive routine could be developed for each of these loading conditions. However, it is sometimes possible to develop a single routine which deals with all possible cases. This is the preferred solution in EPX because it avoids unnecessary duplication of code, simplifies routine updating and maintenance, etc.

In the following, a description of data organization for such generic constitutive routines is given.

2.3 Loading cases

The typical ingredients common to any constitutive routine are the stress tensor σ, the strain increment tensor $\Delta\varepsilon$ and the total strain tensor ε. The latter is typically not used directly in the

constitutive law, if this is formulated in rate form. However, the notion of total strain could be useful e.g. in some failure models of the material, besides of course for post-processing.

In addition to these three tensors, the stress increment tensor $\Delta\sigma$ is also considered sometimes. Of course, all these tensors have the same number of components and the same organization for a given element type.

In the most general (3D continuum) case, each of these tensors has six independent components. For example, for the stress these are σ_x, σ_y, σ_z, τ_{xy}, τ_{yz}, τ_{zx} while for the (total) strain we have ε_x, ε_y, ε_z, γ_{xy}, γ_{yz}, γ_{zx}.

The typical geometrical and loading conditions that can be encountered in EPX are as follows:

- **Bar element (2D or 3D)**. The element responds only in traction / compression. The only non-zero stress component is σ_x, directed along the bar axis. This is sometimes called "uniaxial stress" condition. As concerns strains, the non-zero components are the longitudinal strain ε_x and the two "lateral strain" components ε_y and ε_z (typically with $\varepsilon_y = \varepsilon_z$). Note that these latter two strains cannot be computed from the element geometric configuration (like the other components), but result from the condition of zero lateral stresses in the bar ($\sigma_y = \sigma_z = 0$). Therefore, they (or rather the corresponding strain increments $\Delta\varepsilon_y$, $\Delta\varepsilon_z$) must be computed and returned from the constitutive routine.

- **2D continuum element**. In this case the "transverse shear" components of both stress (τ_{yz}, τ_{zx}) and strain (γ_{yz}, γ_{zx}) are always zero and need not be considered. The remaining four components depend upon the loading condition considered. There are three possibilities: plane stress, plane strain or axisymmetric.

    - In <u>plane stress</u> the "out of plane" normal stress component (z component) is assumed to be zero ($\sigma_z = 0$). Like in the case of the bar, the corresponding strain / strain increment ε_z / $\Delta\varepsilon_z$ has to be computed by the constitutive routine and returned to the element routine. The other three components are generally non-zero.

    - In <u>plane strain</u> the "out of plane" normal strain component is assumed to be zero ($\varepsilon_z = 0$). All other components of strain are non-zero in general, as are all stress components.

    - In the <u>axisymmetric</u> case, all strain and all stress components are non-zero in general.

- **2D beam / conical shell element**. The x component of stress is directed along the element length, like for bars. For uniformity with other shell elements, the z direction is taken along the shell or beam thickness. Note therefore that the element lays on plane xz (and not on plane xy, like it could seem intuitive for a 2D element) so that the "out of plane" direction is y. The "transverse shear" components of both stress (τ_{yz}, τ_{zx}) and strain (γ_{yz}, γ_{zx}) are always zero and need not be considered. Due to beam / shell assumptions, the stress through the element thickness (σ_z) is assumed to be zero. The material routine must therefore always compute the corresponding strain / strain increment ε_z / $\Delta\varepsilon_z$. The in-plane shear components τ_{xy}, γ_{xy} are considered to be zero in the Kirchoff formulation of the beam / shell element (suitable for "thin" elements), while they are non-zero in the Mindlin formulation (suitable for "thick" elements). The component along the element σ_x / ε_x is non-zero in general. Finally the "out of plane" component σ_y / ε_y (also called "hoop" component in the axisymmetric case) depends upon the loading condition considered. There are three possibilities: plane stress, plane strain or axisymmetric.

    - In <u>plane stress</u> the "out of plane" normal stress component is assumed to be zero ($\sigma_y = 0$). Like in the case of the bar, the corresponding strain / strain increment ε_y / $\Delta\varepsilon_y$ has to be computed by the constitutive routine (in addition to ε_z / $\Delta\varepsilon_z$) and returned to the element routine.

- In plane strain the "out of plane" normal strain component is assumed to be zero ($\varepsilon_y = 0$). The corresponding stress / stress increment component must be computed by the constitutive routine.

- In the axisymmetric case, both ε_y and σ_y are non-zero in general. The latter must be computed by the constitutive routine.

- **3D continuum element**. This is the most general case and also the simplest one to treat. All six strain and stress components are non-zero in general. The constitutive routine receives the old stresses and the strain increments, and returns the new stresses.

- **3D beam element**. Due to beam assumptions, the stresses through the element thickness (σ_z) and "out of plane" (σ_y) are assumed to be zero. The material routine must therefore always compute the corresponding strains / strain increments ε_y / $\Delta\varepsilon_y$ and ε_z / $\Delta\varepsilon_z$. The in-plane shear components τ_{xy}, γ_{xy} and the "transverse shear" components of both stress (τ_{yz}, τ_{zx}) and strain (γ_{yz}, γ_{zx}) are considered to be non-zero (otherwise we would have a 3D bar, see first case above, and not a beam). The component along the element σ_x / ε_x is non-zero in general.

- **3D shell element**. Due to shell assumptions, the "out of plane" component of the stress (σ_z) is assumed to be zero. The material routine must therefore always compute the corresponding strain / strain increment ε_z / $\Delta\varepsilon_z$. The "transverse shear" components of both stress (τ_{yz}, τ_{zx}) and strain (γ_{yz}, γ_{zx}) are considered to be zero if the shell element uses the Kirchoff formulation (suitable for "thin" elements), while they are non-zero if the shell uses the Mindlin formulation (suitable for "thick" elements). The other three components of stress and strain are always non-zero in general.

These situations are summarized in the following Table.

Element type	**Case**	**Input to constitutive routine**	**Output from constitutive routine**
Bar (2D or 3D)	N/A	ε_x	σ_x, ε_y, ε_z ($\sigma_y = \sigma_z = 0$)
2D continuum	Plane stress	ε_x, ε_y, γ_{xy}	σ_x, σ_y, ε_z, τ_{xy} ($\sigma_z = 0$)
	Plane strain	ε_x, ε_y, ($\varepsilon_z = 0$), γ_{xy}	σ_x, σ_y, σ_z, τ_{xy}
	Axisymmetric	ε_x, ε_y, ε_z, γ_{xy}	σ_x, σ_y, σ_z, τ_{xy}
2D beam	Plane stress	ε_x, γ_{xy}	σ_x, ε_y, ε_z, τ_{xy} ($\sigma_y = \sigma_z = 0$)
	Plane strain	ε_x, ($\varepsilon_y = 0$), γ_{xy}	σ_x, σ_y, ε_z, τ_{xy} ($\sigma_z = 0$)
Conical shell	Axisymmetric	ε_x, ε_y, γ_{xy}	σ_x, σ_y, ε_z, τ_{xy} ($\sigma_z = 0$)
3D continuum	N/A	ε_x, ε_y, ε_z, γ_{xy}, γ_{yz}, γ_{zx}	σ_x, σ_y, σ_z, τ_{xy}, τ_{yz}, τ_{zx}
3D beam	N/A	ε_x, γ_{xy}, γ_{yz}, γ_{zx}	σ_x, ε_y, ε_z, τ_{xy}, τ_{yz}, τ_{zx} ($\sigma_y = \sigma_z = 0$)
3D shell	N/A	ε_x, ε_y, γ_{xy}, γ_{yz}, γ_{zx}	σ_x, σ_y, ε_z, τ_{xy}, τ_{yz}, τ_{zx} ($\sigma_z = 0$)

The column " input to constitutive routine" shows only the (total) strains for brevity of notation. In reality the input includes also the corresponding (old) stresses and the strain increments, plus of course the state variables and the material constants. Similarly, the column "output from constitutive routine" shows only the (new) stress or (total) strain components, while of course also the associated stress and strain increments, plus the new state variables, must be considered as well.

Highlighted in yellow are any strain (and strain increment) components that must be computed by the constitutive routine. The corresponding stress and stress increment components are assumed to be zero.

In blue are highlighted the shear components (strains in input, stresses in output) which may or may not be present depending on the formulation (Mindlin or Kirchhoff, respectively) assumed for beam and shell elements. When present, such components are independent of any other components and therefore their treatment in the constitutive routine is quite simple.

2.4 Code data structures

The code contains data structures for all the needed properties.

The material properties (constants) which are read from the input data set for each material are stored in a F90 derived type MATERIAL. The field MATERIAL%NBPROP is an INTEGER containing the number of properties (constants) of the material. These are REAL(8) values contained in the MATERIAL%MATVAL(:) field. The type of material is available in MATERIAL%TYPE.

The stress components are contained in a global table called SIG. These quantities vary both in time (time step by time step) and in space (Gauss point by Gauss point). Each element type has a certain number of stress components per Gauss point NBCONG, and a certain number of Gauss points NBGAUL (this latter quantity can be varied by special input directives in case of layered elements, used e.g. to represent composite material structures). Altogether, an element occupies NBCONG*NBGAUL entries in the stress table. An auxiliary table POSIG is used to access the first stress component of each element IEL, which is SIG(POSIG(IEL)).

The total strain components are contained in a table called EPST, which is organized exactly as SIG and uses the same pointer POSIG for addressing purposes.

The hardening quantities (internal variables and any auxiliary material variables) are contained in a global table called ECR. The organization is similar, but not identical, to that of SIG and EPST. In fact, the length of the ECR table depends upon the material type, and this can vary from Gauss point to Gauss point in the same element, in the case of heterogeneous materials, e.g. layered shell or beam elements with a different material in each layer. An auxiliary table POSECR (similar to POSIG) is used to access the first hardening component of each element IEL, which is ECR(POSECR(IEL)).

Normally, only the element routine (or even a higher-level routine) needs to access the global tables. The constitutive routine operates on a Gauss point at a time, and receives in input the portion of the tables “local” to the Gauss point under consideration. Therefore, from the constitutive routine’s perspective, the (local) SIG and EPST tables have a number of entries which depends only upon the type of the element calling the routine, while the (local) ECR table has a fixed number of entries, which depends only upon the current material.

In addition to SIG, EPST and ECR, the constitutive routine exchange list typically contains also the vector of strain increments (DEPS). This vector is a local vector with as many entries as there are stress components in the current element’s Gauss point, which is allocated and computed by the calling (element) routine. In addition, sometimes a similar local vector DSIG of the stress increments is also present, to be computed by the constitutive routine and passed back to the element.

In order to be able to treat all the different cases presented in Section 2.3 by a single constitutive routine (for each material type), the routine must receive from the calling element routine enough additional information to allow identifying the loading case. This information is represented by the following two INTEGER quantities ITAU and IPLANC:

- ITAU is the number of shear components to be considered (both as concerns stresses and strains). As can be seen from the above Table, ITAU can be 0, 1 or 3:
    - ITAU = 0 : there are no shear components.
    - ITAU = 1 : there is one shear component (γ_{xy} and the associated τ_{xy})

  - ITAU = 3 : all shear components are present (γ_{xy} , γ_{yz} , γ_{zx} and the associated τ_{xy} , τ_{yz} , τ_{zx})

- IPLANC is the number of "plane stress" conditions, i.e. the number of normal stress components that are assumed to be zero, which is also the number of normal strain and strain increment components that must be evaluated by the constitutive routine. As can be seen from the above Table, IPLANC can be 0, 1 or 2:

  - IPLANC = 0 : there is no assumed zero normal stress.

  - IPLANC = 1 : there is one assumed zero normal stress. By convention (see Table in Section 2.3) this is always the z component ($\sigma_z = 0$). The constitutive routine must compute and return the corresponding strain increment $\Delta\varepsilon_z$ and total strain ε_z .

  - IPLANC = 2 : there are two assumed zero normal stresses. By convention (see Table in Section 2.3) these are always the y and the z components ($\sigma_y = \sigma_z = 0$). The constitutive routine must compute and return the corresponding strain increments $\Delta\varepsilon_y$, $\Delta\varepsilon_z$ and total strains ε_y , ε_z .

Note, for completeness, that there exists yet another possible value for IPLANC (IPLANC = 3). This does not mean, of course, that there are three assumed zero normal stresses (which would be non-physical) but is used as a flag to activate a special treatment in the constitutive routine, suitable for some shell elements developed at CEA (more details on this below).

Summarizing, the exchange list of a typical solid material routine is as follows:

```
SUBROUTINE MY_MAT (ITAU, IPLANC, XMAT, SIG, EPST, DEPS, ECR, NEWSIG, …)
```

where:

- ITAU, IPLANC allow to uniquely identify the loading case as explained above. These are used but not modified in the routine.

- XMAT(*) is the table of material constants for the present material, These are used but not modified in the routine.

- SIG(*) is the old stress table for the current Gauss point, i.e. the stress components at the beginning of the current time step. These are used but not modified in the routine.

- EPST(*) is the new total strain table for the current Gauss point, i.e. the total strain components at the end of the current time step. If some of the strain increment components are computed inside the constitutive routine (i.e. if IPLANC =1 or 2) then the corresponding values of total strain are also updated in the constitutive routine.

- DEPS(*) is the table of strain increments for the current Gauss point, over the current time step. If some of the strain increment components are computed inside the constitutive routine (i.e. if IPLANC =1 or 2) then the corresponding values of total strain are also updated in the constitutive routine.

- ECR(*) is the table of hardening (or internal) variables for the current Gauss point. In input, these are the old values (at the beginning of the time step). They are updated inside the constitutive routine and therefor upon return they are the new values (at the end of the time step).

- NEWSIG(*) is the new stress table for the current Gauss point, i.e. the stress components at the end of the current time step. These are computed by the constitutive routine and returned back to the element routine. Note that, even when present, NEWSIG is not guaranteed to be different from SIG. In fact, some element routines pass the same table both as SIG and as NEWSIG. This means that in the constitutive routine, once a certain component of NEWSIG

has been assigned to, the corresponding component of SIG (which in theory should contain the old stress) is no longer guaranteed to be available, and therefore should not be used.

- … indicates a set of additional data which may or may not be present, depending on the specific material type. For example, to deal with element erosion one would typically need the index of the current Gauss point and the total number of Gauss points for this element.

Note that the NEWSIG table is not always present in the exchange list. Sometimes (i.e., for some material types) it is replaced by a DSIG table, containing the stress increments as computed and returned by the constitutive routine. In this case the new stresses can be obtained in the element routine as the old ones plus the increments. Other times neither NEWSIG nor DSIG are present, and in this case the SIG table is directly updated (modified) in the constitutive routine. Of course, it would be advisable to uniformize as far as possible these exchange lists, in order to simplify code reading and to limit possible errors.

2.5 Programming conventions and rules

All tables in exchange list are declared with a variable (unknown) dimension in the constitutive routine, as indicated by the asterisk. For example:

```
REAL(8), INTENT(IN) :: XMAT(*)
```

To simplify programming as far as possible, the following conventions are tacitly assumed:

- The length of the XMAT and ECR tables depend only upon the material type. Since we are in the material routine, the lengths should be well known to the programmer, who is responsible for avoiding any memory-related programming errors: no value beyond the true length of the table should be used or, even worse, modified, since this would destroy the memory.

- The length of all other tables (SIG, EPST, DEPS, NEWSIG, DSIG) depends in principle upon the calling element type and upon the loading case. However, only two cases are considered in EPX. The number of components can only be either 4 or 6. The former case corresponds to ITAU = 0 or 1, while the latter case corresponds to ITAU = 3.

The second convention should simplify the programming of constitutive routines. In fact, the programmer can assume that the first four components of stress and strain are always available, while the last two (transverse shears) are only available when ITAU = 2.

Note that, since an asterisk is used in the declaration, no run-time error would arise from incorrect addressing of the "forbideen" components, even by using a debugger. Thus, the following programming rule must be respected:

- When ITAU = 0 or 1, the components 5 and 6 of SIG and the other similar tables must neither be used nor be assigned to, since they are simply not allocated.

- Otherwise (ITAU = 3) the components 5 and 6 of SIG and the other similar tables can and must be both used and assigned to.

2.6 Components organization

In principle the programmer of a constitutive routine is free to choose any organization for the stress and strain components. This may or may not correspond with the organization assumed by the (calling) element routine.

If the two organizations do not match, in EPX it is the element routine which is responsible of taking appropriate actions. In practice, a conversion routine is called from the element routine just before calling the constitutive routine, so that the tables are re-organized in the way expected by the material. Then the material routine is called. Finally, a second conversion routine is called to bring back the (updated) tables to the organization expected by the element.

The vast majority of element routines in EPX adopt one of the following two organizations for the stress and strain components:

Component	Organization 1	Organization 2	Notes
1	xx	xx	
2	yy	yy	
3	zz	xy	zz and xy swapped
4	xy	zz	zz and xy swapped
5	yz	yz	Only in 3D
6	zx	zx	Only in 3D

Organization 1 is typically used by CEA's elements in 3D, while organization 2 is typically used by CEA's elements in 2D and by JRC's elements in both 2D and 3D. In order to pass from one of these two organizations to the other, the conversion routine simply has to swap components 3 and 4 of the stress and strain table.

3 Resume of relations for elasticity

We recall some expressions used in linear elasticity.

3.1 3D continuum case

The linear (infinitesimal) **strain tensor** is given by:

$$\boldsymbol{\varepsilon} = \left[\varepsilon_{ij} \right] = \begin{bmatrix} \varepsilon_{xx} & \varepsilon_{xy} & \varepsilon_{xz} \\ \varepsilon_{yx} & \varepsilon_{yy} & \varepsilon_{yz} \\ \varepsilon_{zx} & \varepsilon_{zy} & \varepsilon_{zz} \end{bmatrix} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \tag{1}$$

with $\mathbf{u} = [u_i] = [u_x \quad u_y \quad u_z]$ the displacement field. The tensor is symmetric, i.e. $\varepsilon_{ij} = \varepsilon_{ji}$ (with $i \neq j$). Usually, the normal strain components (diagonal terms) are indicated by a single index: $\varepsilon_i = \varepsilon_{ii}$ (no sum on i). For the shear components (off-diagonal terms), usually the "engineering" shear strains $\gamma_{ij} = 2\varepsilon_{ij}$ (with $i \neq j$) are considered instead of the strain tensor components ε_{ij} :

$$\gamma_{ij} = \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \quad \text{(with } i \neq j\text{)} \tag{2}$$

Thus, the strain can be written in vector notation as:

$$\boldsymbol{\varepsilon} = [\varepsilon_x \quad \varepsilon_y \quad \varepsilon_z \quad \gamma_{xy} \quad \gamma_{yz} \quad \gamma_{zx}]^T \tag{3}$$

The Cauchy stress tensor is given by:

$$\boldsymbol{\sigma} = \left[\sigma_{ij} \right] = \begin{bmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{bmatrix} \tag{4}$$

The tensor is symmetric, i.e. $\sigma_{ij} = \sigma_{ji}$ (with $i \neq j$). Usually, the normal stress components (diagonal terms) are indicated by a single index: $\sigma_i = \sigma_{ii}$ (no sum on i). For the shear components usually the notation τ_{ij} is used instead of σ_{ij} . Thus, the stress can be written in **Voigt notation** as:

$$\boldsymbol{\sigma} = [\sigma_x \quad \sigma_y \quad \sigma_z \quad \tau_{xy} \quad \tau_{yz} \quad \tau_{zx}]^T \tag{5}$$

In linear elasticity, the stress and the strain are related by Hooke's law:

$$\boldsymbol{\sigma} = \mathbf{D}\boldsymbol{\varepsilon} \tag{6}$$

or, in components

$$\sigma_{ij} = D_{ijkl}\varepsilon_{kl} \tag{7}$$

where $\mathbf{D} = [D_{ijkl}]$ is a fourth-order tensor which is function of the actual material properties.

For an **isotropic** linear elastic material it can be shown that only two material constants suffice to completely determine $\mathbf{D}$. For example, by considering **Young's modulus** E and **Poisson's coefficient** ν one has (by assuming that both $\boldsymbol{\varepsilon}$ and $\boldsymbol{\sigma}$ are written in Voigt notation, i.e. with 6 independent components each):

$$\mathbf{D} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix} \tag{8}$$

Alternatively, one can use **Lamé's constants**, defined as:

$$\lambda = \frac{\nu E}{(1+\nu)(1-2\nu)} \qquad \text{and} \qquad \mu = G \tag{9}$$

with G the **shear modulus**, which is related to E and ν by:

$$G = \frac{E}{2(1+\nu)} \tag{10}$$

In this way the expression of $\mathbf{D}$ becomes:

$$\mathbf{D} = \begin{bmatrix} \lambda+2\mu & \lambda & \lambda & 0 & 0 & 0 \\ \lambda & \lambda+2\mu & \lambda & 0 & 0 & 0 \\ \lambda & \lambda & \lambda+2\mu & 0 & 0 & 0 \\ 0 & 0 & 0 & \mu & 0 & 0 \\ 0 & 0 & 0 & 0 & \mu & 0 \\ 0 & 0 & 0 & 0 & 0 & \mu \end{bmatrix} \tag{11}$$

or, in components:

$$D_{ijkl} = \lambda\delta_{ij}\delta_{kl} + \mu(\delta_{ik}\delta_{jl} + \delta_{il}\delta_{jk}) \tag{12}$$

so that the stress-strain relation can also be written:

$$\sigma_{ij} = \lambda \varepsilon_{kk} \delta_{ij} + 2\mu \varepsilon_{ij} \tag{13}$$

with $\varepsilon_{kk} = \varepsilon_{xx} + \varepsilon_{yy} + \varepsilon_{zz} = \varepsilon_x + \varepsilon_y + \varepsilon_z$ the trace (first invariant) of the strain tensor, also called the **volumetric strain** e:

$$e = \varepsilon_x + \varepsilon_y + \varepsilon_z \tag{14}$$

The relations between the strains and the stresses (strain-stress relations) can be written:

$$\begin{aligned}
\varepsilon_x &= \frac{1}{E}[\sigma_x - \nu(\sigma_y + \sigma_z)] \\
\varepsilon_y &= \frac{1}{E}[\sigma_y - \nu(\sigma_x + \sigma_z)] \\
\varepsilon_z &= \frac{1}{E}[\sigma_z - \nu(\sigma_x + \sigma_y)] \\
\gamma_{xy} &= \frac{1}{G}\tau_{xy} \\
\gamma_{yz} &= \frac{1}{G}\tau_{yz} \\
\gamma_{zx} &= \frac{1}{G}\tau_{zx}
\end{aligned} \tag{15}$$

while the inverse (stress-strain) relations are (several alternative forms are shown):

$$\begin{aligned}
\sigma_x &= \frac{\nu E}{(1+\nu)(1-2\nu)} e + \frac{E}{1+\nu}\varepsilon_x = \lambda e + 2\mu\varepsilon_x = (\lambda + 2\mu)\varepsilon_x + \lambda(\varepsilon_y + \varepsilon_z) \\
\sigma_y &= \frac{\nu E}{(1+\nu)(1-2\nu)} e + \frac{E}{1+\nu}\varepsilon_y = \lambda e + 2\mu\varepsilon_y = (\lambda + 2\mu)\varepsilon_y + \lambda(\varepsilon_x + \varepsilon_z) \\
\sigma_z &= \frac{\nu E}{(1+\nu)(1-2\nu)} e + \frac{E}{1+\nu}\varepsilon_z = \lambda e + 2\mu\varepsilon_z = (\lambda + 2\mu)\varepsilon_x + \lambda(\varepsilon_x + \varepsilon_y) \\
\tau_{xy} &= G\gamma_{xy} = 2G\varepsilon_{xy} = 2\mu\varepsilon_{xy} \\
\tau_{yz} &= G\gamma_{yz} = 2G\varepsilon_{yz} = 2\mu\varepsilon_{yz} \\
\tau_{xz} &= G\gamma_{xz} = 2G\varepsilon_{xz} = 2\mu\varepsilon_{xz}
\end{aligned} \tag{16}$$

By denoting Θ the sum of the normal stresses, i.e. the trace (first invariant) of the stress tensor $\sigma_{kk} = \sigma_{xx} + \sigma_{yy} + \sigma_{zz} = \sigma_x + \sigma_y + \sigma_z$:

$$\Theta = \sigma_x + \sigma_y + \sigma_z \tag{17}$$

and by adding together the first three strain-stress equations (15) above one obtains:

$$e = \frac{1-2\nu}{E}\Theta \tag{18}$$

If a body is subjected to a uniform hydrostatic pressure p we have:

$$\sigma_x = \sigma_y = \sigma_z = -p \tag{19}$$

so that by adding together the first three equations (15) one gets:

$$e = -\frac{3(1-2\nu)}{E} p = -\frac{1}{K} p \tag{20}$$

The coefficient:

$$K = \frac{E}{3(1-2\nu)} \tag{21}$$

is called **bulk modulus**. Other expressions of K as a function of λ and μ or of Θ and e are:

$$K = \lambda + \frac{2}{3}\mu \quad \text{or} \quad K = \frac{\Theta}{3e} \tag{22}$$

3.2 Uniaxial stress

The uniaxial stress case (e.g. a simple traction or compression along the axial direction of a bar assumed to be along x by convention) is characterized by zero lateral stresses:

$$\sigma_y = \sigma_z = 0 \tag{23}$$

and by lateral strains resulting from Poisson's effect:

$$\varepsilon_y = \varepsilon_z = -\nu\varepsilon_x \tag{24}$$

where ε_x is the longitudinal strain. The longitudinal stress σ_x is simply given by:

$$\sigma_x = E\varepsilon_x \tag{25}$$

In fact, by replacing (24) into the stress-strain relations (16) and by using the definitions (9) and (10) of Lamé's constants we obtain (25) and (23):

$$\begin{aligned}
\sigma_x &= (\lambda + 2\mu)\varepsilon_x + \lambda(-\nu\varepsilon_x - \nu\varepsilon_x) = \\
&= (\lambda + 2\mu)\varepsilon_x - 2\lambda\nu\varepsilon_x = \\
&= (\lambda + 2\mu - 2\lambda\nu)\varepsilon_x = \\
&= [\lambda(1-2\nu) + 2\mu]\varepsilon_x = \\
&= \left(\frac{\nu E}{1+\nu} + \frac{E}{1+\nu}\right)\varepsilon_x = E\varepsilon_x
\end{aligned} \tag{26}$$

$$\begin{aligned}
\sigma_y &= -\nu(\lambda + 2\mu)\varepsilon_x + \lambda\varepsilon_x - \lambda\nu\varepsilon_x = \\
&= (-\lambda\nu - 2\mu\nu + \lambda - \lambda\nu)\varepsilon_x = \\
&= [(1-2\nu)\lambda - 2\mu\nu]\varepsilon_x = \\
&= \left(\frac{\nu E}{1+\nu} - \frac{E\nu}{1+\nu}\right)\varepsilon_x = 0
\end{aligned} \tag{27}$$

$$\sigma_z = -\nu(\lambda + 2\mu)\varepsilon_x + \lambda\varepsilon_x - \lambda\nu\varepsilon_x = 0 \tag{28}$$

3.3 Uniaxial strain

The uniaxial strain case corresponds to traction or compression along one direction (x by convention) by inhibiting the lateral strains. Therefore:

$$\varepsilon_y = \varepsilon_z = 0 \tag{29}$$

By replacing (29) into the stress-strain relations (16) one gets:

$$\begin{aligned} \sigma_x &= (\lambda + 2\mu)\varepsilon_x \\ \sigma_y &= \sigma_z = \lambda \varepsilon_x \end{aligned} \tag{30}$$

From these one obtains the following relation between the lateral stress and the longitudinal stress:

$$\sigma_y = \sigma_z = \frac{\lambda}{\lambda + 2\mu}\sigma_x \tag{31}$$

By using the definitions (9) and (10) of Lamé's constants, the first of (30) becomes:

$$\begin{aligned} \sigma_x &= (\lambda + 2\mu)\varepsilon_x = \\ &= [\frac{\nu E}{(1+\nu)(1-2\nu)} + \frac{E}{(1+\nu)}]\varepsilon_x = \\ &= \frac{\nu E + (1-2\nu)E}{(1+\nu)(1-2\nu)}\varepsilon_x = \\ &= \frac{1-\nu}{(1+\nu)(1-2\nu)} E\varepsilon_x \end{aligned} \tag{32}$$

By comparing (32) with (25) we see that, for a given longitudinal strain, the longitudinal stress is larger in uniaxial strain than in uniaxial stress. From a one-dimensional point of view, it is like if the material would have a larger apparent Young's modulus:

$$E_{\text{apparent}} = \frac{1-\nu}{(1+\nu)(1-2\nu)} E \tag{33}$$

The coefficient multiplying E on the right-hand side of (33) is 1.0 for $\nu = 0$, grows with ν and tends to infinity for $\nu \to 0.5$ (incompressible material).

3.4 Plane stress

If a body is very thin, i.e. if its size along one of the axes (z by convention) is very small compared with the other two sizes (in the plane xy by convention), and if the loading and boundary conditions on the body do not depend on z, then one may assume plane stress condition, which is characterized by:

$$\sigma_z = 0 \tag{34}$$

Correspondingly, the strain along the z direction can be computed from the third strain-stress relation (15) as (by assuming linear elastic behaviour):

$$\varepsilon_z = -\frac{\nu}{E}(\sigma_x + \sigma_y) \tag{35}$$

or also:

$$\varepsilon_z = -\frac{\lambda}{\lambda + 2\mu}(\varepsilon_x + \varepsilon_y) \tag{36}$$

In addition, also the "transverse shear" stress (and strain components) may or may not be considered as zero:

$$\begin{aligned} \gamma_{yz} &= \gamma_{zx} = 0 \\ \tau_{yz} &= \tau_{zx} = 0 \end{aligned} \tag{37}$$

This is the case, for example in 2D plane stress analysis. In 3D shells, however, the vanishing of transverse shear components is only assumed in Kirchhoff's theory (suitable for very thin shells), while it is not assumed in Mindlin's theory (suitable for moderately thick shells).

3.5 Plane strain

If a body is very thick, i.e. if its size along one of the axes (z by convention) is very large compared with the other two sizes (in the plane xy by convention), or if a body is glued between two rigid plates normal to the z axis whose distance is kept constant, and if the loading and boundary conditions on the body do not depend on z , then one may assume plane strain condition, which is characterized by:

$$\varepsilon_z = 0 \tag{38}$$

Correspondingly, the stress along the z direction can be computed from the third strain-stress relation (15) as (by assuming linear elastic behaviour):

$$\sigma_z = \nu(\sigma_x + \sigma_y) \tag{39}$$

3.6 Deviatoric stress tensor

The **deviatoric stress tensor** is defined as:

$$s_{ij} = \sigma_{ij} - \frac{1}{3}\sigma_{kk}\delta_{ij} \tag{40}$$

or, in matrix form:

$$\mathbf{s} = \left[s_{ij} \right] = \begin{bmatrix} \sigma_{xx} - \frac{1}{3}\sigma_{kk} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} - \frac{1}{3}\sigma_{kk} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} - \frac{1}{3}\sigma_{kk} \end{bmatrix} \tag{41}$$

Denoting π the **mean normal stress**:

$$\pi = \frac{1}{3}\sigma_{kk} = \frac{\sigma_{xx} + \sigma_{yy} + \sigma_{zz}}{3} \tag{42}$$

we may rewrite (40) as:

$$\sigma_{ij} = s_{ij} + \pi\delta_{ij} \tag{43}$$

i.e. the Cauchy stress tensor $\boldsymbol{\sigma}$ can be expressed as the sum of two other stress tensors:

- The **mean normal stress tensor** (or **volumetric stress tensor**) $\boldsymbol{\Pi}$ which thends to **change the volume** of the stressed body (without causing any distortion):

$$\boldsymbol{\Pi} = \left[\Pi_{ij}\right] = \begin{bmatrix} \pi & 0 & 0 \\ 0 & \pi & 0 \\ 0 & 0 & \pi \end{bmatrix} \tag{44}$$

- The deviatoric stress tensor $\mathbf{s}$ given by (41), which tends to **distort** the stressed body (without causing any volume change).

3.7 Principal stresses

Let $\boldsymbol{\sigma}$ as given by (4) be the Cauchy stress tensor at a generic point P of a body. The components of the tensor are referred to the global axes (x, y, z) and represent the forces per unit area acting on the faces of an infinitesimal cube with a vertex in P and aligned with the axes x, y, z.

If we consider a generic plane of unit normal $\mathbf{n}$ passing through point P, we can obtain the stress (force per unit surface) $\mathbf{t}_n$ acting on the plane (a vector that in general is not normal to the plane, i.e. it is not aligned with $\mathbf{n}$) by considering an infinitesimal tetrahedron with a vertex in P, having three faces normal to the global axes x, y, z and the fourth one normal to $\mathbf{n}$. Equilibrium of this tetrahedron to translation yields:

$$\boldsymbol{\sigma}^T \mathbf{n} = \mathbf{t}_n \tag{45}$$

Since $\boldsymbol{\sigma}$ is symmetric, $\boldsymbol{\sigma}^T = \boldsymbol{\sigma}$ and (45) can also be written:

$$\boldsymbol{\sigma}\mathbf{n} = \mathbf{t}_n \tag{46}$$

We want to find under which circumstances (i.e., for which particular planes through P, that is for which particular orientations of $\mathbf{n}$) the stress is directed along the normal, i.e.:

$$\mathbf{t}_n = s\mathbf{n} \tag{47}$$

where the scalar s is the magnitude of the stress vector acting on the plane. These particular stresses are denoted the **principal stresses** at P and the corresponding normals are denoted the **principal directions**. By comparing (47) and (46) we obtain:

$$\mathbf{t}_n = \boldsymbol{\sigma}\mathbf{n} = s\mathbf{n} \tag{48}$$

which can also be written:

$$(\boldsymbol{\sigma} - s\mathbf{I})\mathbf{n} = \mathbf{0} \tag{49}$$

having indicated with $\mathbf{I}$ the identity (or unit) tensor and with $\mathbf{0}$ the zero vector:

$$\mathbf{I} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \qquad \mathbf{0} = \begin{bmatrix} 0 & 0 & 0 \end{bmatrix}^T \tag{50}$$

By expanding (49):

$$\begin{aligned}(\sigma_{xx}-s)n_x+\tau_{xy}n_y+\tau_{xz}n_z&=0\\ \tau_{yx}n_x+(\sigma_{yy}-s)n_y+\tau_{yz}n_z&=0\\ \tau_{zx}n_x+\tau_{zy}n_y+(\sigma_{zz}-s)n_z&=0\end{aligned}\tag{51}$$

The linear algebraic system (51) admits solutions $[n_i]=\mathbf{n}$ not all identically zero (the trivial solution), and such must be the n_i which, representing direction cosines, must also satisfy the condition $n_i n_i=1$, if and only if the determinant of (51) vanishes, i.e. if:

$$\begin{vmatrix}(\sigma_{xx}-s) & \tau_{xy} & \tau_{xz}\\ \tau_{yx} & (\sigma_{yy}-s) & \tau_{yz}\\ \tau_{zx} & \tau_{zy} & (\sigma_{zz}-s)\end{vmatrix}=0\tag{52}$$

By expanding the eterminant in (52) we obtain a cubic algebraic equation in s (which can be considered as a parameter):

$$-s^3+As^2-Bs+C=0\tag{53}$$

with:

$$\begin{aligned}A&=\sigma_{xx}+\sigma_{yy}+\sigma_{zz}\\ B&=\begin{vmatrix}\sigma_{xx} & \tau_{xy}\\ \tau_{yx} & \sigma_{yy}\end{vmatrix}+\begin{vmatrix}\sigma_{xx} & \tau_{xz}\\ \tau_{zx} & \sigma_{zz}\end{vmatrix}+\begin{vmatrix}\sigma_{yy} & \tau_{yz}\\ \tau_{zy} & \sigma_{zz}\end{vmatrix}=\\ &=\sigma_{xx}\sigma_{yy}+\sigma_{xx}\sigma_{zz}+\sigma_{yy}\sigma_{zz}-\tau_{xy}^2-\tau_{yz}^2-\tau_{zx}^2\\ C&=\begin{vmatrix}\sigma_{xx} & \tau_{xy} & \tau_{xz}\\ \tau_{yx} & \sigma_{yy} & \tau_{yz}\\ \tau_{zx} & \tau_{zy} & \sigma_{zz}\end{vmatrix}=\sigma_{xx}\sigma_{yy}\sigma_{zz}-\sigma_{xx}\tau_{yz}^2-\sigma_{yy}\tau_{xz}^2-\sigma_{zz}\tau_{xy}^2\end{aligned}\tag{54}$$

Since the coefficients of (53) are real, its three roots (say s_1, s_2, s_3), the **eigenvalues**, are also real and can be obtained by solving (53) by any suitable method. By replacing one eigenvalue s_i at a time in the system (51) and by solving for n_i, one finds the three (mutually orthogonal) principal directions, the **eigenvectors**, which satisfy the stress normality condition (47).

By rewriting (51) referred to the principal directions, we obtain:

$$\begin{vmatrix}(s_1-s) & 0 & 0\\ 0 & (s_2-s) & 0\\ 0 & 0 & (s_3-s)\end{vmatrix}=0\tag{55}$$

from which one sees immediately that the three roots are s_1, s_2 and s_3.

By expanding the determinant (55) like we have done above for (52) we obtain the cubic equation (53) in the form:

$$-s^3+A's^2-B's+C'=0\tag{56}$$

with:

$$\begin{aligned} A' &= s_1 + s_2 + s_3 \\ B' &= s_1 s_2 + s_1 s_3 + s_2 s_3 \\ C' &= s_1 s_2 s_3 \end{aligned} \tag{57}$$

Since the equations (53) and (56) must have the same roots (the s_i) the coefficients must be equal, i.e.:

$$A' = A \qquad B' = B \qquad C' = C \tag{58}$$

and, since the orientation x, y, z of the global axes was completely arbitrary, we may conclude that the expressions A, B, C are **invariant**, i.e. their values do not change by changing the reference frame. They are called the **first**, **second** and **third invariant of the stress tensor**, respectively:

$$\begin{aligned} I_1 &= A = A' \\ I_2 &= B = B' \\ I_3 &= C = C' \end{aligned} \tag{59}$$

3.8 Stress invariants

As seen in the previos paragraph, the **stress** tensor (4) has the following three **invariants** (given hereafter with several alternative expressions).

$$\begin{aligned} I_1 &= \sigma_{ii} \\ &= \sigma_{xx} + \sigma_{yy} + \sigma_{zz} \\ &= \text{tr}\ \boldsymbol{\sigma} \\ I_2 &= \frac{1}{2}(\sigma_{ii}\sigma_{jj} - \sigma_{ij}\sigma_{ji}) \\ &= \sigma_{xx}\sigma_{yy} + \sigma_{yy}\sigma_{zz} + \sigma_{xx}\sigma_{zz} - \sigma_{xy}^2 - \sigma_{yz}^2 - \sigma_{zx}^2 \\ &= \begin{vmatrix} \sigma_{yy} & \sigma_{yz} \\ \sigma_{zy} & \sigma_{zz} \end{vmatrix} + \begin{vmatrix} \sigma_{xx} & \sigma_{xz} \\ \sigma_{zx} & \sigma_{zz} \end{vmatrix} + \begin{vmatrix} \sigma_{xx} & \sigma_{xy} \\ \sigma_{yx} & \sigma_{yy} \end{vmatrix} \\ &= \frac{1}{2}[(\text{tr}\ \boldsymbol{\sigma})^2 - \text{tr}\ (\boldsymbol{\sigma}\boldsymbol{\sigma})] \\ I_3 &= \frac{1}{3}\sigma_{ij}\sigma_{jk}\sigma_{ki} \\ &= \sigma_{xx}\sigma_{yy}\sigma_{zz} + 2\sigma_{xy}\sigma_{yz}\sigma_{zx} - \sigma_{xy}^2\sigma_{zz} - \sigma_{yz}^2\sigma_{xx} - \sigma_{zx}^2\sigma_{yy} \\ &= |\boldsymbol{\sigma}| \\ &= \det \boldsymbol{\sigma} \\ &= \frac{1}{6}[(\text{tr}\ \boldsymbol{\sigma})^3 - 3\ \text{tr}\ \boldsymbol{\sigma}\ \text{tr}\ \boldsymbol{\sigma}^2 + 2\ \text{tr}\ \boldsymbol{\sigma}^3] \end{aligned} \tag{60}$$

where tr denotes the trace of the tensor, i.e. the sum of the diagonal terms, and $|\ |$ or det denotes the determinant.

Note that I_1 has the dimensions of a stress [Pa], I_2 of a stress square [Pa^2] and I_3 of a stress cube [Pa^3]. These quantities are objective, i.e. they do not vary upon rigid rotation of the coordinate system.

By denoting σ_1, σ_2, σ_3 the **principal stresses**, and by noting that in the principal reference all shear components vanish, the invariants can also be expressed as follows:

$$\begin{aligned} I_1 &= \sigma_1 + \sigma_2 + \sigma_3 \\ I_2 &= \sigma_1\sigma_2 + \sigma_2\sigma_3 + \sigma_3\sigma_1 \\ I_3 &= \sigma_1\sigma_2\sigma_3 \end{aligned} \tag{61}$$

In analogy with (60) and (61), the three **invariants** of the **deviatoric stress tensor** are (several alternative forms are shown):

$$\begin{aligned} J_1 &= s_{kk} = 0 \\ J_2 &= \frac{1}{2} s_{ij} s_{ji} \\ &= \frac{1}{6}[(\sigma_{xx} - \sigma_{yy})^2 + (\sigma_{yy} - \sigma_{zz})^2 + (\sigma_{zz} - \sigma_{xx})^2] + \sigma_{xy}^2 + \sigma_{yz}^2 + \sigma_{zx}^2 \\ &= \frac{1}{3}(\sigma_{xx}^2 + \sigma_{yy}^2 + \sigma_{zz}^2 - \sigma_{xx}\sigma_{yy} - \sigma_{yy}\sigma_{zz} - \sigma_{zz}\sigma_{xx}) + \sigma_{xy}^2 + \sigma_{yz}^2 + \sigma_{zx}^2 \\ J_3 &= \frac{1}{3} s_{ij} s_{jk} s_{ki} \end{aligned} \tag{62}$$

Because $s_{kk} = J_1 = 0$, the deviatoric stress tensor represents a state of pure shear. It can be shown that the principal directions of the deviatoric stress tensor $\mathbf{s}$ coincide with those of the stress tensor $\boldsymbol{\sigma}$. By denoting s_1, s_2, s_3 the **principal deviatoric stresses**, one has also:

$$\begin{aligned} J_2 &= \frac{1}{2}(s_1^2 + s_2^2 + s_3^2) \\ J_3 &= s_1 s_2 s_3 \end{aligned} \tag{63}$$

Furthermore, J_2 can be expressed as a function of the principal stresses:

$$J_2 = \frac{1}{6}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_2)^2] \tag{64}$$

Finally, the deviatoric stress invariants can be expressed as a function of the stress invariants:

$$\begin{aligned} J_2 &= \frac{1}{3} I_1^2 - I_2 \\ J_3 &= \frac{2}{27} I_1^3 - \frac{1}{3} I_1 I_2 + I_3 \end{aligned} \tag{65}$$

3.9 Equivalent or von Mises stress

The **equivalent stress** or **von Mises stress** is defined as:

$$\begin{aligned}\sigma_{\text{eq}} &= \sqrt{3J_2} \\ &= \sqrt{\frac{3}{2}s_{ij}s_{ij}} \\ &= \sqrt{\frac{1}{2}[(\sigma_{xx}-\sigma_{yy})^2+(\sigma_{yy}-\sigma_{zz})^2+(\sigma_{zz}-\sigma_{xx})^2]+3\sigma_{xy}^2+3\sigma_{yz}^2+3\sigma_{zx}^2} \\ &= \sqrt{\frac{1}{2}[(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_2)^2]}\end{aligned} \tag{66}$$

This can be used to predict **yielding** of materials under **multiaxial loading** conditions, by using results from simple **uniaxial tensile tests**.

References

1. EUROPLEXUS User's Manual, on-line version: http://europlexus.jrc.ec.europa.eu.

European Commission
EUR 27683 EN – Joint Research Centre – Institute for the Protection and Security of the Citizen

Title: Some notes on the organization of routines for solid materials in EUROPLEXUS

Author(s): Folco Casadei, Georgios Valsamos, Martin Larcher, Vegard Aune

Luxembourg: Publications Office of the European Union

2015 – 22 pp. – 21.0 x 29.7 cm

EUR – Scientific and Technical Research series – ISSN 1831-9424 (online)

ISBN 978-92-79-54259-6 (PDF)

doi:10.2788/98573

Abstract

This document describes the organization of some routines for the modelling of solid materials in the EUROPLEXUS computer code (EPX) at the Joint Research Centre (JRC) of the European Commission (EC) at Ispra.

LB-NA-27683-EN-N

JRC Mission

As the Commission's in-house science service, the Joint Research Centre's mission is to provide EU policies with independent, evidence-based scientific and technical support throughout the whole policy cycle.

Working in close cooperation with policy Directorates-General, the JRC addresses key societal challenges while stimulating innovation through developing new methods, tools and standards, and sharing its know-how with the Member States, the scientific community and international partners.

Serving society
Stimulating innovation
Supporting legislation

Publications Office

ISBN 978-92-79-54259-6
doi:10.2788/98573