

# manual material editor

# 8

id  
impact  
design  
europe

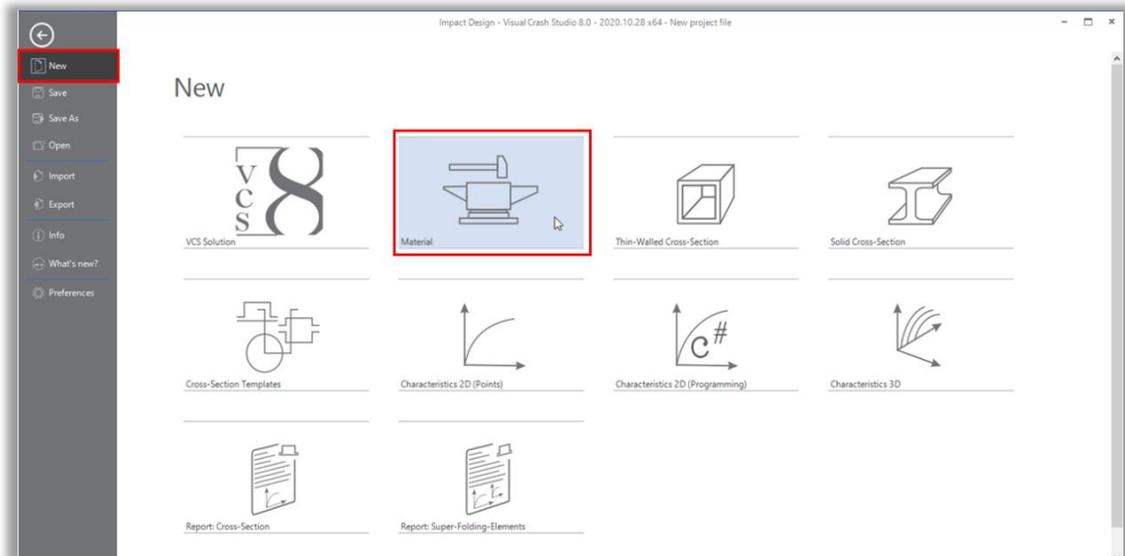
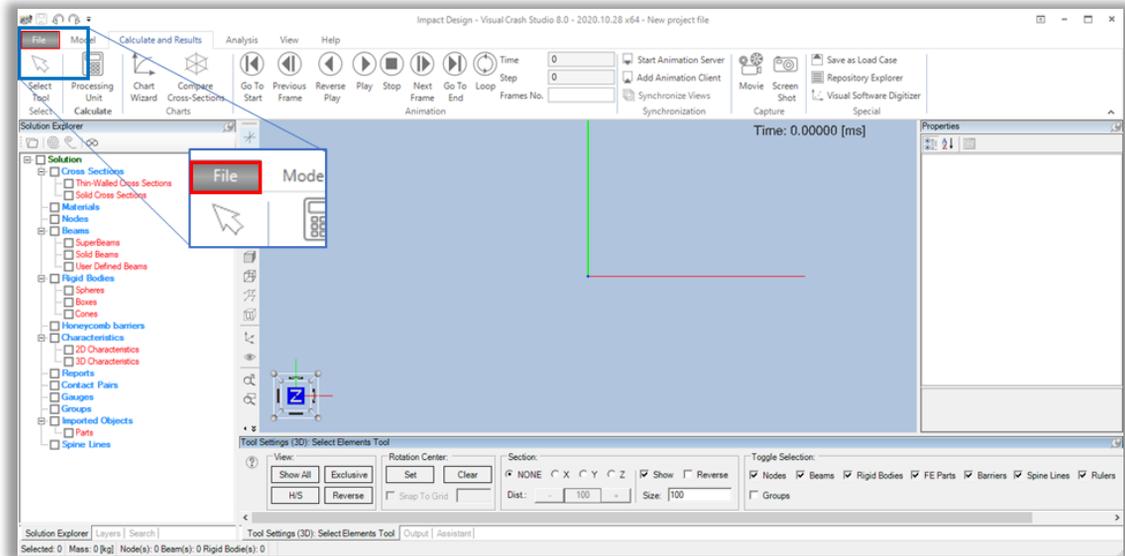


*The fastest way in Pre Design*

➤ <b><u>NEW MATERIAL</u></b>	<b>3</b>
➤ <b><u>MATERIAL CONSTANTS</u></b>	<b>4</b>
➤ <b><u>STRESS STRAIN CHARACTERISTIC TYPE</u></b>	<b>5</b>
▪ <u>ARRAY - 2D ARRAY REPRESENTATION</u>	5
▪ <u>POWER LAW</u>	6
▪ <u>POLYNOMIAL REPRESENTATION</u>	6
▪ <u>USER FUNCTION (ANALYTICAL)</u>	7
▪ <u>2D CHARACTERISTIC</u>	7
▪ <u>USER FUNCTION (DISCRETE)</u>	9
▪ <u>ARRAY 3D</u>	10
▪ <u>SATURATION TYPE</u>	11
➤ <b><u>STRAIN HARDENING TYPE</u></b>	<b>12</b>
▪ <u>MIXED HARDENING RESPONSE</u>	13
▪ <u>SATURATION TYPE RESPONSE</u>	13
➤ <b><u>STRAIN RATE CHARACTERISTIC</u></b>	<b>14</b>
▪ <u>COWPER SYMONDS</u>	14
▪ <u>MODIFIED COWPER SYMONDS 1</u>	15
▪ <u>MODIFIED COWPER SYMONDS 2</u>	15
▪ <u>THE USER DEFINED FUNCTION – 3D CHARACTERISTIC</u>	16
▪ <u>JOHNS COOK</u>	16
➤ <b><u>HARDENING FACTOR</u></b>	<b>17</b>
➤ <b><u>FRACTURE INDICATOR</u></b>	<b>19</b>
▪ <u>SURFACE STRAIN</u>	19
▪ <u>CL</u>	20
➤ <b><u>THE MATERIAL AND CHARACTERISTIC REPOSITORY</u></b>	<b>21</b>
➤ <b><u>VISUAL SOFTWARE DIGITIZER</u></b>	<b>23</b>
➤ <b><u>MATERIAL AND CHARACTERISTIC ENCRYPTION</u></b>	<b>25</b>
➤ <b><u>SORT MATERIALS BY NAME</u></b>	<b>27</b>
➤ <b><u>REMOVE DUPLICATED MATERIALS</u></b>	<b>28</b>
➤ <b><u>REMOVE UNUSED MATERIALS</u></b>	<b>28</b>

# NEW MATERIAL

The commands sequence: *File* → *New* → *Material* creates an empty *Material* object on the *Materials Branch* of the *Solution Explorer* tree and opens the *Material Editor* window.



The *Material* is described by five groups of data in the *Properties* window:

- [Material Constants](#)
- [Stress Strain Characteristic](#)
- [Strain Rate Characteristic](#)
- [Hardening Factor](#)
- [Fracture Indicator](#)



# MATERIAL CONSTANTS

The Material Constant section of the Properties window defines standard Strength of Material data: Hardening Factor, Mass Density, Poisson Ratio, Proof Strain, Proof Stress, Young Modulus.

- **Hardening Factor** - Contribution of Isotropic / Kinematic Hardening; Hardening Factor describes fractional contribution of kinematic/isotropic contributions to the total hardening response of Material (see also 'Hardening Factor')
- **Mass Density** - [kg/m<sup>3</sup>]
- **Poisson Ratio** - [1]
- **Proof Strain** - [1] typically 0,2% (0.002)
- **Proof Stress** - initial yield stress in [MN/m<sup>2</sup>] at proof strain (typically 0,2%)
- **Young Modulus** - [MN/m<sup>2</sup>]

Properties	
1. Material - Attributes	
Type	DuctileIsotropic
2. Material Constants	
HardeningFactor	1
MassDensity	7890
PoissonRatio	0.3
ProofStrain	0.002
ProofStress	380
YoungModulus	200000
3. Stress - Strain Characteristic	
4. Strain - Rate Effects	
5. Fracture Indicator	
Other	
Comment	
Guid	97934834-0803-4333-9f95-d69dfa03e425
Layer	Default
Name	Example MATERIAL

# STRESS STRAIN CHARACTERISTIC

The Material Constant section of the Properties window defines standard Strength of Material data: Hardening Factor, Mass Density, Poisson Ratio, Proof Strain, Proof Stress, Young Modulus.

- **Hardening Factor** - Contribution of Isotropic / Kinematic Hardening; Hardening Factor describes fractional contribution of kinematic/isotropic contributions to the total hardening response of Material (see also 'Hardening Factor')
- **Mass Density** - [kg/m<sup>3</sup>]
- **Poisson Ratio** - [1]
- **Proof Strain** - [1] typically 0,2% (0.002)
- **Proof Stress** - initial yield stress in [MN/m<sup>2</sup>] at proof strain (typically 0,2%)
- **Young Modulus** - [MN/m<sup>2</sup>]

Properties	
1. Material - Attributes	
Type	DuctileIsotropic
2. Material Constants	
HardeningFactor	1
MassDensity	7890
PoissonRatio	0.3
ProofStrain	0.002
ProofStress	380
YoungModulus	200000
3. Stress - Strain Characteristic	
StressStrainCharacteristic	Array: [10] points defined
StressStrainCharacteristicType	Array
StressStrainHardeningLaw	100 [%] isotropic
StressStrainHardeningType	Isotropic
StressStrainMeasure	True
4. Strain - Rate Effects	
5. Fracture Indicator	
Other	
Comment	
Guid	97934834-0803-4333-9f95-d69dfa03e425
Layer	Default
Name	Example MATERIAL



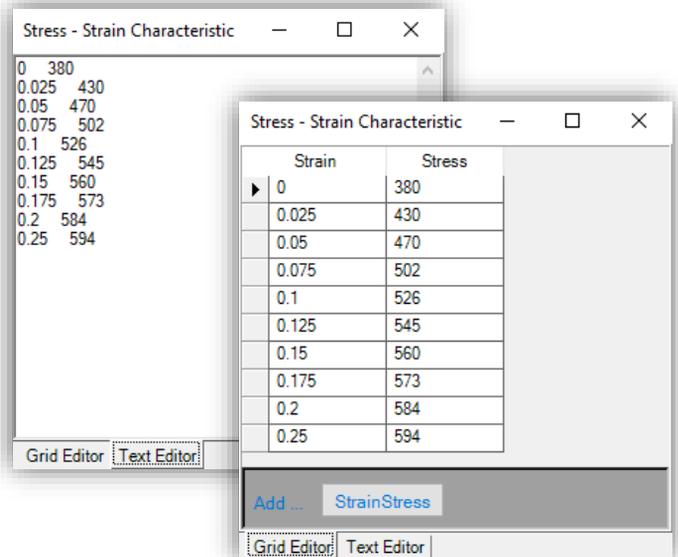
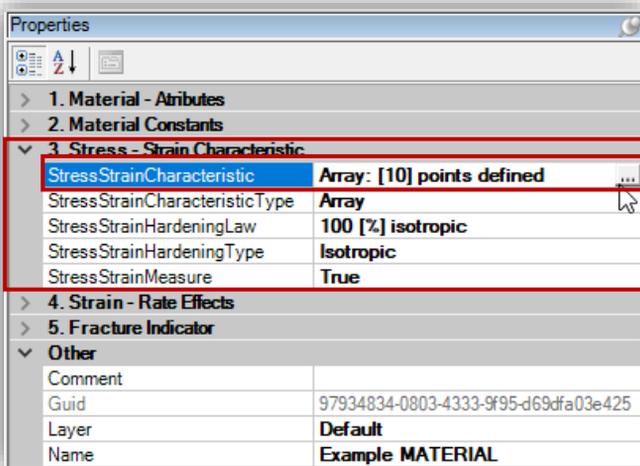
# STRESS STRAIN CHARACTERISTIC TYPE

In the Stress Strain Characteristic Type field in the Properties window you can find the drop-down list containing five methods of definition of stress – strain relation:

- Array
- Power Law
- Polynomial
- User Function
- Array 3D
- Saturation Type

## ARRAY – 2D ARRAY REPRESENTATION

In the 2D Array representation model the Stress – Strain response in the plastic range is defined as an array of discrete strain – stress points.



After clicking on the '...' button in the Stress Strain Characteristic field in the Properties window the Stress-Strain Characteristic window will open. In this window you will find grid and text editor for the array definition.

The syntax is: one Point per line, x and y coordinates separated by TAB.

Next line – press 'Enter'

The Strain and Stress can be defined either in engineering or true measures. It is important to correctly define the actual measure in the Stress Strain Measure field in the Properties window, by entering the appropriate option (True or Engineering).

The strain defined in this section is treated as an actual plastic strain (total strain minus proof strain). The strain must be given in standard fractional units (not in %). The stress must be defined in MPa (MN/m<sup>2</sup>).



# STRESS STRAIN CHARACTERISTIC TYPE

## POWER LAW

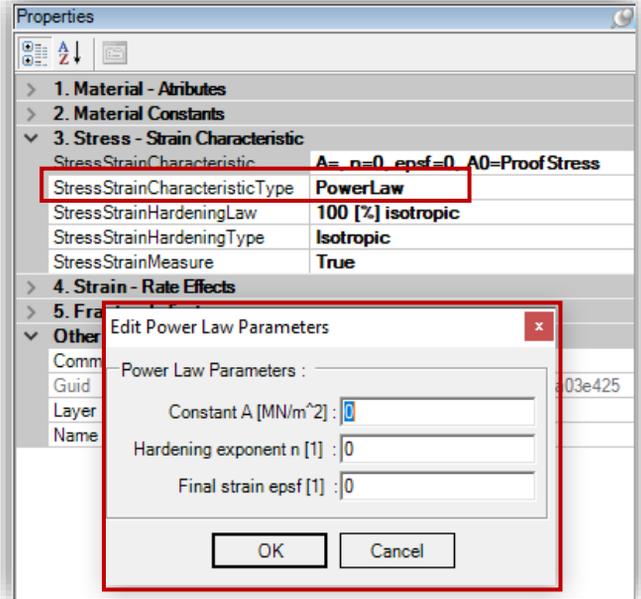
After selecting the Power Law model in the Stress Strain Characteristic Type field in the Properties window you can define the stress – strain curve.

Analytical representation of the stress-strain curve can be defined in several ways. The elementary definition results from the power-type law:

$$\sigma(\varepsilon) = \sigma_y + A\varepsilon^n$$

This law is implemented explicitly in VCS as Power Law model of the Stress-Strain Characteristic.

After clicking on the '...' button in the Stress Strain Characteristic field in the Properties window the 'Edit Power Law Parameters' window will appear. In this window you can define the Constant A, hardening exponent n and the final strain.



## POLYNOMIAL REPRESENTATION

After selecting the Polynomial model in the Stress Strain Characteristic Type field in the Properties window you can define the stress – strain curve.

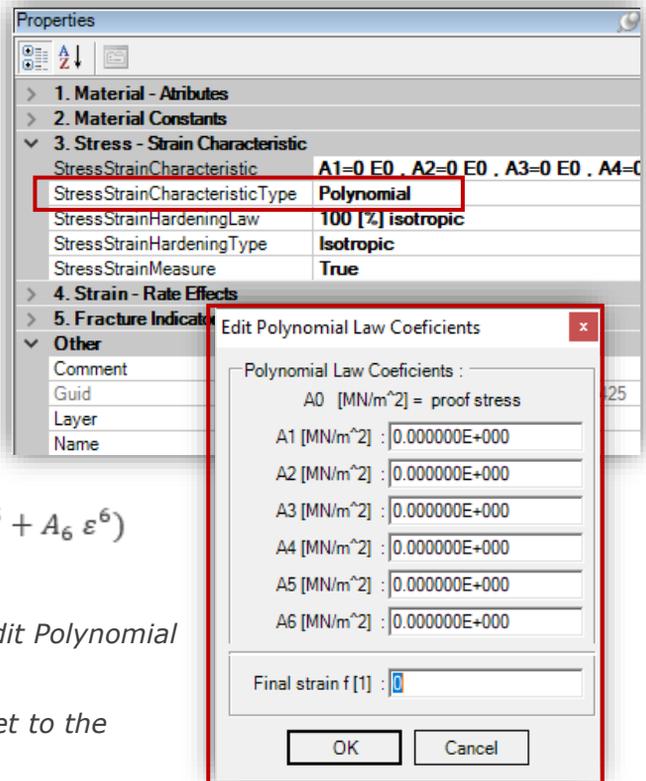
The stress-strain relation can be given as a polynomial – up to the 6-th order:

$$\sigma(\varepsilon) = \sigma_y + (A_1 \varepsilon + A_2 \varepsilon^2 + A_3 \varepsilon^3 + A_4 \varepsilon^4 + A_5 \varepsilon^5 + A_6 \varepsilon^6)$$

After clicking on the '...' button in the Stress Strain Characteristic field in the Properties window the 'Edit Polynomial Law Coefficients' window will appear.

Note, that by default the 0-th order coefficient is set to the proof stress.

Remaining coefficients are defined in the Polynomial Law Editor.

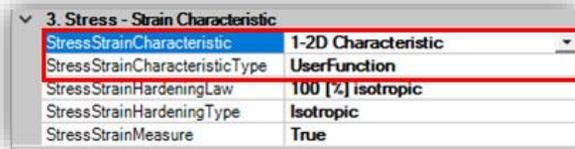


# STRESS STRAIN CHARACTERISTIC TYPE

## USER FUNCTION (ANALYTICAL)

Any programmable representation of the stress strain relation

$$\sigma=f(\varepsilon)$$



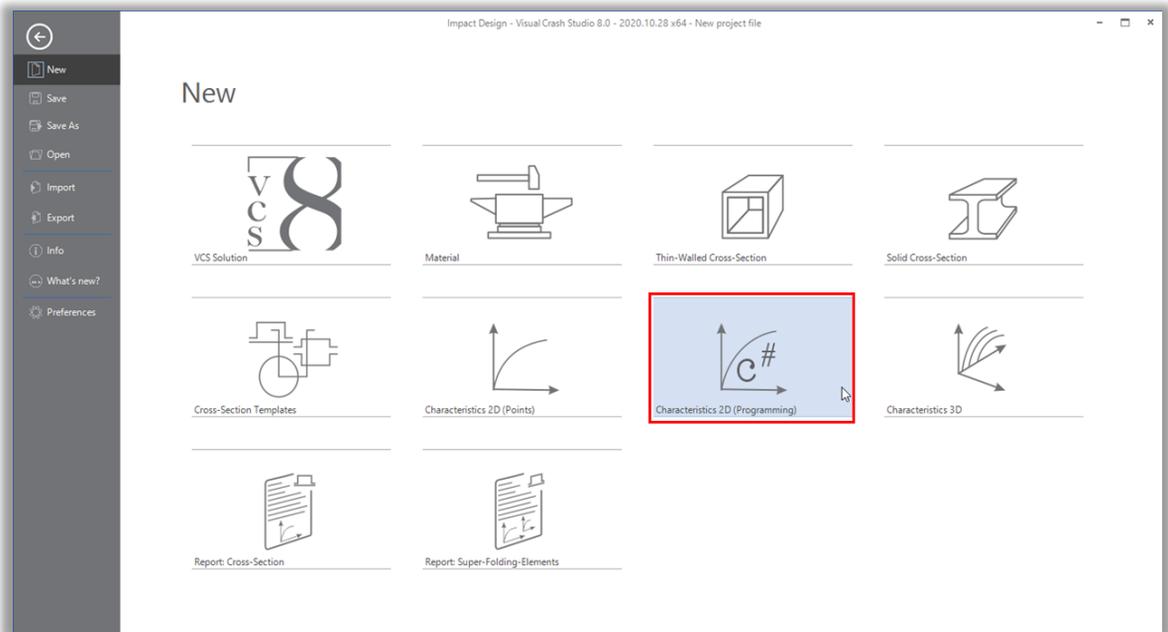
3. Stress - Strain Characteristic	
StressStrainCharacteristic	1-2D Characteristic
StressStrainCharacteristicType	UserFunction
StressStrainHardeningLaw	100 [%] isotropic
StressStrainHardeningType	Isotropic
StressStrainMeasure	True

Can be defined by means of the 2D Characteristic Editor

## 2D CHARACTERISTIC

The commands sequence: File → New → Characteristics 2D (Programming)

creates an empty Characteristic on the Characteristics Branch of the Solution Explorer tree and opens the Characteristic Editor – 1-2D Characteristic window.



The body of the one-independent-variable function  $f(x)$  must be defined by the user in C# language. Note that the domain of the function (in this case maximal strain to fracture) is defined separately.

The 2D characteristic specified in the Stress Strain Characteristic field defines stress-strain relation.

The  $x$  and  $y$  variables in  $y = f(x)$  function correspond to strain and stress, respectively.

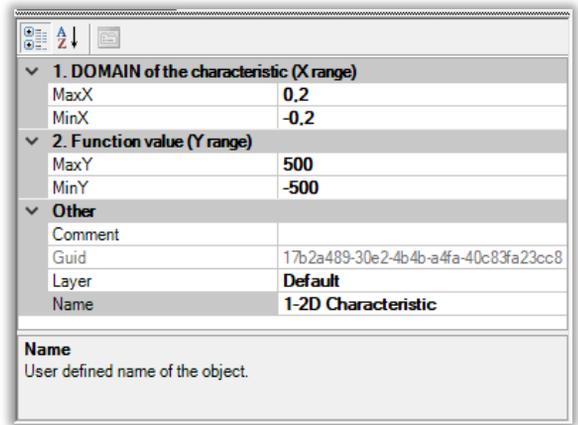
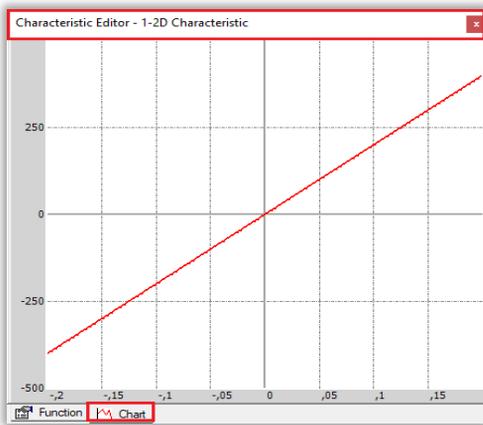
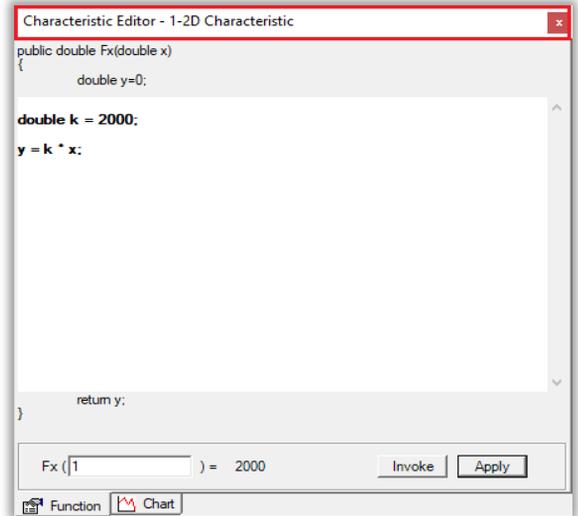
The example code on the right corresponds to the power law representation of stress strain and gives results analogous to the Power Law model of the Stress Strain Characteristic Type



# 2D CHARACTERISTIC

The user function utility is equipped with an elementary debugger that reports compilation error(s). It is recommended to check the correctness of function definition by calculating the function value for a meaningful independent variable.

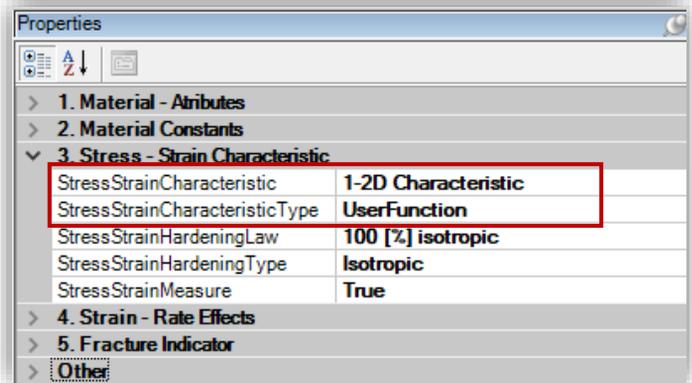
This can be done in the bottom part of the Characteristic Editor. The function value is calculated by pressing the 'Invoke' button. Current definition of the function is implemented once the 'Apply' button is pressed. The diagram of newly defined function can now be displayed by clicking on the 'Chart' tab



## Reminder:

The limits of dependent and independent variables must be defined by the user in the Properties window. Depending on the characteristic implementation these limits may have different meanings. In the case of stress-strain definition the Max X type defines the final (rupture) strain of the material, while Min X must be set to zero.

Finally, the user function can be assigned to the Material by selecting the 2D Characteristic name from the drop-down list in the Stress Strain Characteristic field in the Material Properties window.



## Reminder:

Note that characteristics defined in the VCS can be assigned to various objects. Therefore, it is important to keep meaningful names of characteristic objects and store them in clearly organized folder system.

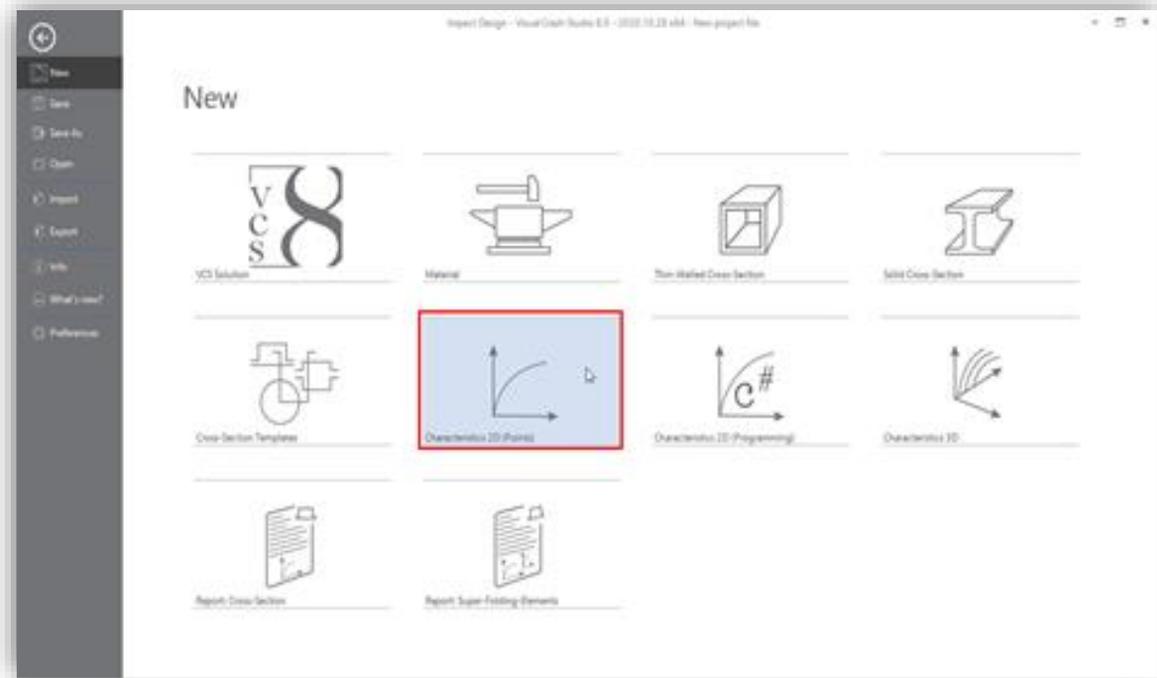


# USER FUNCTION (DISCRETE) – 2D CHARACTERISTIC

The 2D characteristic can also be defined by specifying list of discrete values of stress-strain points.

The commands sequence: *File* → *New* → *Characteristic 2D (Points)*

creates an empty Characteristic on the Characteristics Branch of the Solution Explorer tree and opens the Characteristic Editor – 0-2D Characteristic window.



In this window you can define the list of points.

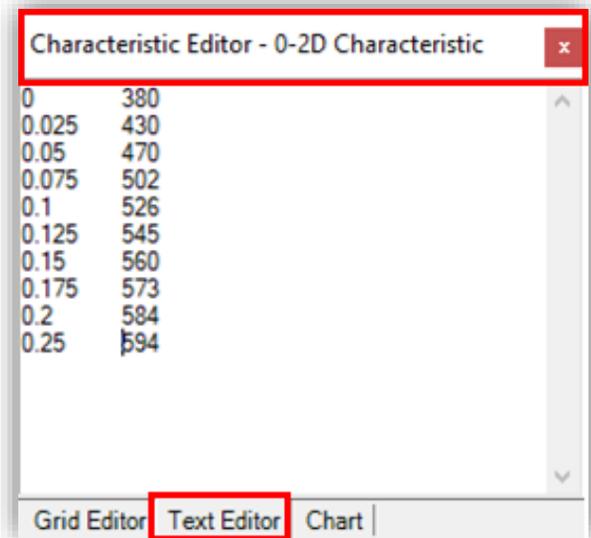
The syntax is: one Point per line, x and y coordinates separated by TAB.

Next line – press 'Enter'.

The corresponding (interpolated) characteristic is displayed by selecting the Chart tab In the Editor window.

The Point List method is analogous to the 2D Array representation but is much slower in execution. Therefore, it is not recommended in material representation procedures.

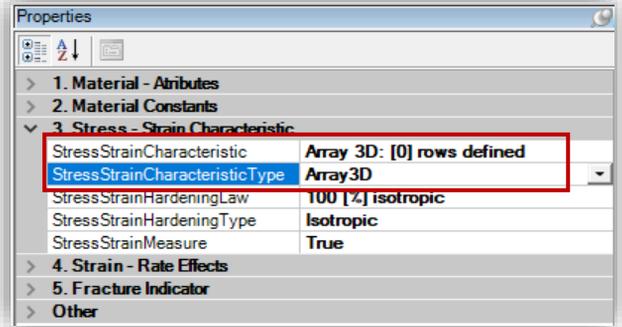
*Note that similarly like in the case of C# characteristic the domain of strains must be defined separately in properties window*



# STRESS STRAIN CHARACTERISTIC TYPE

## ARRAY 3D

Array3D type of stress-strain-strain rate representation facilitates direct transfer of experimental strain-strain rate-stress tabular data directly into the VCS Material.

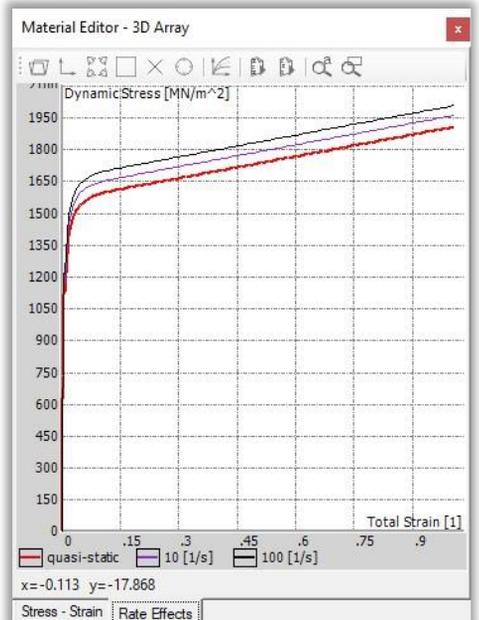


	0.004	1	20	200
0	1116	1151.74	1193.91	1247.24
0.01	1389.33	1425.07	1467.24	1520.58
0.02	1468.35	1504.09	1546.26	1599.6
0.03	1512.38	1548.12	1590.29	1643.63
0.04	1539.64	1575.38	1617.55	1670.88
0.05	1557.58	1593.32	1635.49	1688.83
0.06	1570.02	1605.76	1647.93	1701.27
0.07	1579.11	1614.84	1657.02	1710.35
0.08	1586.1	1621.84	1664.02	1717.35
0.09	1591.8	1627.53	1669.71	1723.04
0.1	1596.66	1632.4	1674.57	1727.9
0.12	1605.01	1640.74	1682.92	1736.25
0.14	1612.46	1648.19	1690.37	1743.7
0.16	1619.55	1655.28	1697.46	1750.79
0.18	1626.49	1662.23	1704.4	1757.73
0.2	1633.37	1669.11	1711.28	1764.61
0.3	1667.6	1703.34	1745.51	1798.84
0.4	1701.8	1737.54	1779.71	1833.04
0.5	1736	1771.74	1813.91	1867.24
0.6	1770.2	1805.94	1848.11	1901.44
0.7	1804.4	1840.14	1882.31	1935.64
0.8	1838.6	1874.34	1916.51	1969.84
0.9	1872.8	1908.54	1950.71	2004.04
1	1907	1942.74	1984.91	2038.24

	0.004	1	20	200
0	1116	1151.74	1193.91	1247.24
0.01	1389.33	1425.07	1467.24	1520.58
0.02	1468.35	1504.09	1546.26	1599.6
0.03	1512.38	1548.12	1590.29	1643.63
0.04	1539.64	1575.38	1617.55	1670.88
0.05	1557.58	1593.32	1635.49	1688.83
0.06	1570.02	1605.76	1647.93	1701.27
0.07	1579.11	1614.84	1657.02	1710.35
0.08	1586.1	1621.84	1664.02	1717.35
0.09	1591.8	1627.53	1669.71	1723.04
0.1	1596.66	1632.4	1674.57	1727.9
0.12	1605.01	1640.74	1682.92	1736.25
0.14	1612.46	1648.19	1690.37	1743.7
0.16	1619.55	1655.28	1697.46	1750.79
0.18	1626.49	1662.23	1704.4	1757.73
0.2	1633.37	1669.11	1711.28	1764.61
0.3	1667.6	1703.34	1745.51	1798.84
0.4	1701.8	1737.54	1779.71	1833.04
0.5	1736	1771.74	1813.91	1867.24
0.6	1770.2	1805.94	1848.11	1901.44
0.7	1804.4	1840.14	1882.31	1935.64
0.8	1838.6	1874.34	1916.51	1969.84
0.9	1872.8	1908.54	1950.71	2004.04
1	1907	1942.74	1984.91	2038.24

Perhaps the most effective procedure of the table definition is a direct copy/paste transfer of the table from the Microsoft Office Excel program.

The quasi-static (plastic) stress-strain relation and dynamic response of the defined Material can be displayed by selecting appropriate tabs (Stress - Strain or Rate Effect) on the Material Editor window.



### Reminder:

In order to open the Material Editor window double click on the selected Material in the Solution Explorer window



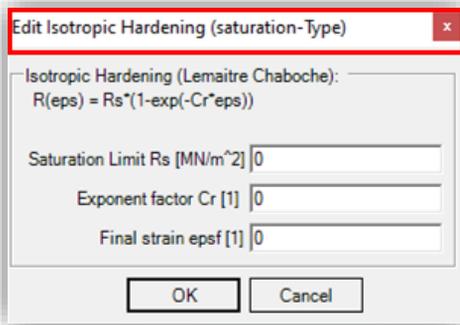
# STRESS STRAIN CHARACTERISTIC TYPE

## SATURATION TYPE (THE LEMAITRE / CHABOCHE FORMULATION)

In the Lemaitre / Chaboche formulation both kinematic and isotropic contributions are defined as a saturation-type laws:

$$R(\epsilon) = R_S (1 - e^{-C_r \epsilon}) \quad \text{and} \quad \alpha(\epsilon) = \alpha_S (1 - e^{-C_a \epsilon})$$

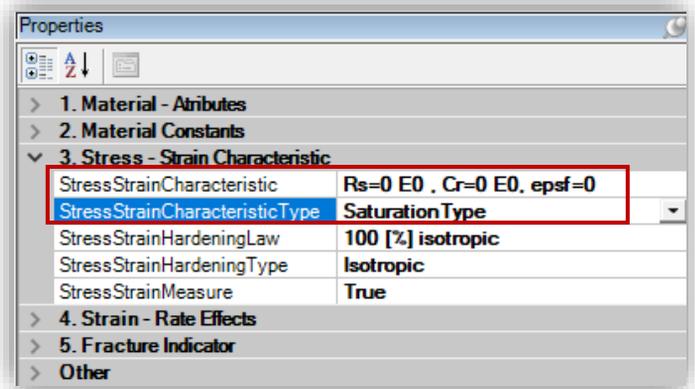
where  $R_S$  and  $\alpha_S$  denote, respectively, saturation limits ( $\epsilon \rightarrow \infty$ ) for isotropic and kinematic hardening, while  $C_r$  and  $C_a$  are two scalar material constants.



The isotropic hardening contribution,  $R(\epsilon)$ , is defined in the Stress Strain Characteristic section, when the Saturation Type hardening law is defined in Stress Strain Characteristic Type drop-down list.

After clicking on the '...' button in the Stress Strain Characteristic field in the Properties window the Edit Isotropic Hardening (saturation type) window will open.

The isotropic contribution,  $\alpha(\epsilon)$ , is defined separately in the Strain Hardening Type section of the Properties window. If not defined its contribution to the total hardening is assumed to be zero.



# STRAIN HARDENING TYPE

In the Stress Strain Hardening Type field in the Properties window you can find the drop-down list containing four methods of defining the stress-strain material response on loading, unloading and reverse loading paths.

- Isotropic
- Kinematic
- Mixed Scalar
- Saturation Type

VCS offers two methods for accounting on the stress-strain material response on loading, unloading, reverse loading paths.

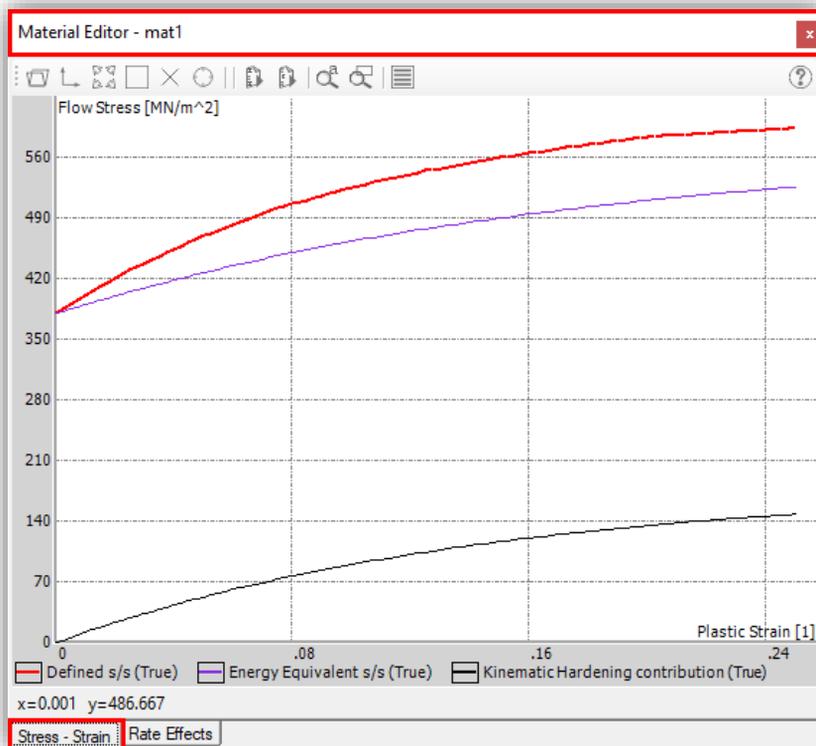
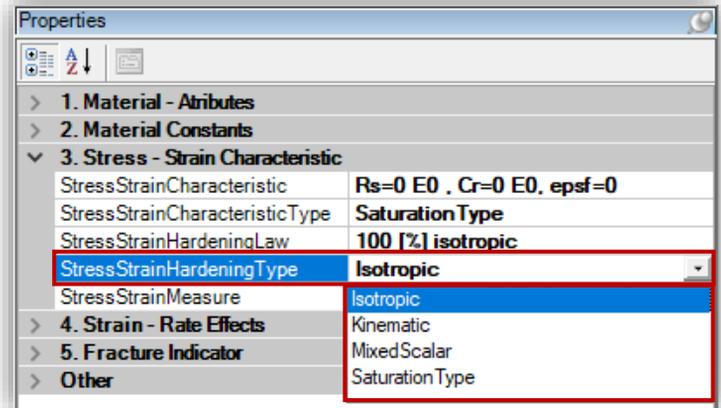
The first method is purely experimental and involves physical or virtual experiments on axial crushing of prismatic members. This methodology is based on the hardening factor concept

The second method applies directly material law with proper definition of kinematic/isotropic strain hardening contributions to the Super Folding Element calculation routines. Both methods can be used jointly. In this case hardening factor has a meaning of a pure experimental scaling factor.

Two types of kinematic/isotropic hardening laws are available in VCS:

- Mixed (scalar) hardening law and
- Saturation-type law (Lemaitre / Chaboche model)

When **kinematic hardening contribution** for a given Material is defined graph corresponding to this contribution is shown on the stress strain characteristic graph. Black curve shows contribution of the kinematic hardening to total strain hardening response of material.



# STRAIN HARDENING TYPE

## MIXED HARDENING RESPONSE

In VCS the general representation of static strain hardening is given as:

$$\sigma(\epsilon) = \sigma_y^0 + K(\epsilon)$$

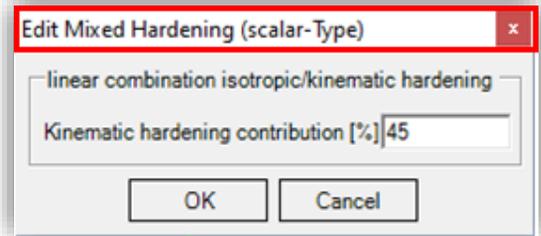
where  $K(\epsilon)$  denotes total strain hardening as a function of plastic strain,  $\epsilon$ , while  $\sigma_y^0$  denotes initial yield stress of the material (typically 0.2% proof stress). In the one-scalar parameter formulation the total strain hardening of material  $K(\epsilon)$  is expressed as a linear combination of isotropic and kinematic hardening defined by single parameter  $\eta$ :

$$K(\epsilon) = (1 - \eta)K(\epsilon) + \eta K(\epsilon)$$

$$\text{where } R(\epsilon) \stackrel{\text{def}}{=} (1 - \eta)K(\epsilon) \quad \text{and} \quad \alpha(\epsilon) \stackrel{\text{def}}{=} \eta K(\epsilon)$$

The hardening parameter is defined by selecting the Mixed Scalar type in the Stress Strain Hardening Type section of the material property window. An actual value of the hardening parameter is defined in Stress Hardening Law section.

Selection of Isotropic type automatically sets  $\eta$  to zero while selection of Kinematic type sets  $\eta=1$ . Intermediate values are defined in a dialog window that opens from Stress Hardening Law section.



## SATURATION TYPE RESPONSE (THE LEMAITRE / CHABOCHE FORMULATION)

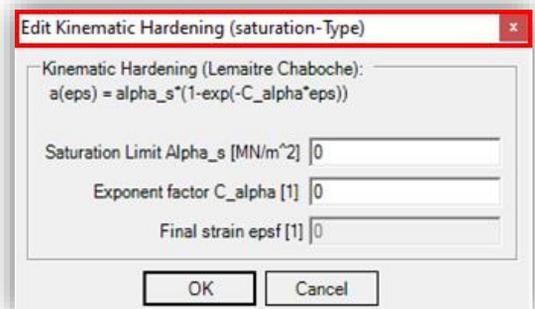
In the Lemaitre / Chaboche formulation both, kinematic and isotropic contributions are defined as a saturation-type laws:

$$R(\epsilon) = R_S (1 - e^{-C_r \epsilon}) \quad \text{and} \quad \alpha(\epsilon) = \alpha_S (1 - e^{-C_\alpha \epsilon})$$

where  $R_S$  and  $\alpha_S$  denote, respectively, saturation limits ( $\epsilon \rightarrow \infty$ ) for isotropic and kinematic hardening, while  $C_r$  and  $C_\alpha$  are two scalar material constants.

The kinematic hardening contribution  $\alpha(\epsilon)$ , is defined in the Stress Hardening Law section when the Saturation Type hardening law is defined in Stress Strain Hardening Type drop-down list.

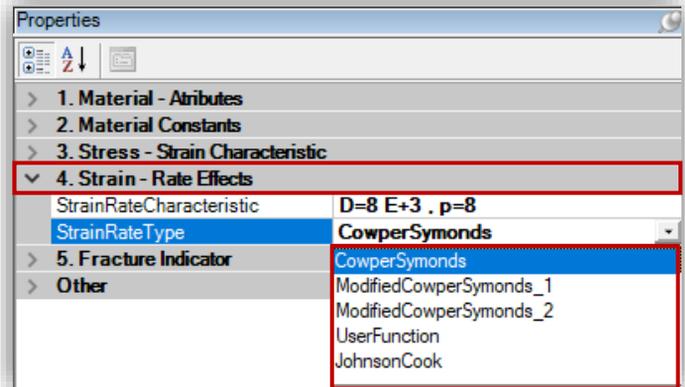
The isotropic contribution is defined separately in the Stress Strain Characteristic section. If not defined kinematic contribution to the total hardening is assumed to be zero



# STRAIN RATE CHARACTERISTIC

The Strain - Rate Effect section of the Properties window offers five different representations of the dynamic factor:

- Cowper Symonds
- Modified Cowper Symonds 1
- Modified Cowper Symonds 2
- The user defined function
- Johnson Cook



The elementary definition follows the well-known Cowper-Symonds definition that originally described the strain rate sensitivity of the yield (or proof) stress.

The Cowper Symonds law was later extended to more complex material response subjected to large plastic strains.

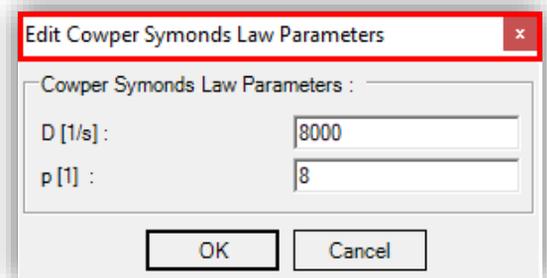
Typically, two sets of Cowper-Symonds type relations are defined in order to represent strain rate sensitivity of two characteristic stress values on the complex stress-strain-strain rate characteristic. Most common choice is the proof stress and Ultimate Tensile Stress (UTS) in the engineering measures.

Few attempts have been made in the published reports to define a non-linear combination of characteristic Cowper Symonds relations that would cover the entire strain-strain rate domain. It appears, however, that a general law – applicable to a wide class of materials – does not exist. Therefore, two types of nonlinear combination of Cowper Symonds law, suggested by Abramowicz, are implemented in VCS in order to provide the user with flexible tool for dynamic factor definition. Finally, the User Function type allows for implementation of any programmable dynamic factor function of two independent variables.

## COWPER SYMONDS

After selecting the Cowper Symonds representation of the dynamic factor in the Strain Rate Type field of the Properties window, when clicking on the '...' button in the Stress Strain Characteristic field the Edit Coper Symonds Law Parameters window will open.

The classic two parameters of the Cowper Symonds power law



$$(\sigma(\epsilon, \dot{\epsilon}))' / (\sigma^{\wedge'}(\epsilon)) = 1 + [(\dot{\epsilon}' / D)]^{\wedge(1/p)}$$

is implemented as Cowper Symonds type in Strain Rate Effects section of the Material properties. The material constants D and p are defined in the text box editor (Edit Coper Symonds Law Parameters window)



# STRAIN RATE CHARACTERISTIC

## MODIFIED COWPER SYMONDS 1

After selecting the Modified Cowper Symonds 1 representation of the dynamic factor in the Strain Rate Type field of the Properties window, when clicking on the '...' button in the Stress Strain Characteristic field the Edit Coper Symonds Law Parameters window will open.

The Modified Cowper Symonds Law implements a nonlinear combination of simple power type relations defined for proof stress,  $D_y$ , and UTS,  $D_u$ :

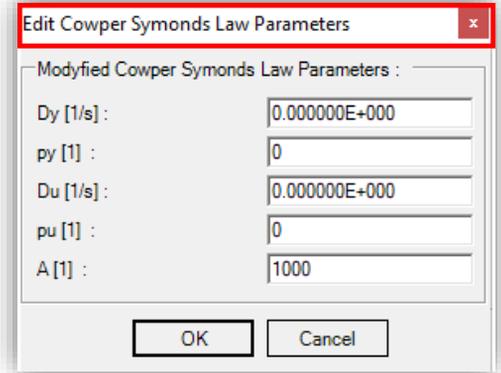
$$D_y(\dot{\epsilon}) \frac{\sigma_y(\epsilon, \dot{\epsilon})}{\sigma_y^s(\epsilon)} = 1 + \left(\frac{\dot{\epsilon}_y}{D_y}\right)^{1/p_y}$$

$$D_u(\dot{\epsilon}) \frac{\sigma_u(\epsilon, \dot{\epsilon})}{\sigma_u^s(\epsilon)} = 1 + \left(\frac{\dot{\epsilon}_u}{D_u}\right)^{1/p_u}$$

The fast-decaying exponential function is taken as 'coupling' factor in the Modified Cowper Symonds 1 type

$$\frac{\sigma(\epsilon, \dot{\epsilon})}{\sigma^s(\epsilon)} = [D_y(\dot{\epsilon}) - D_u(\dot{\epsilon})]e^{-\epsilon A} + D_u(\dot{\epsilon})$$

The material parameters of the rate function are defined in the corresponding editor.



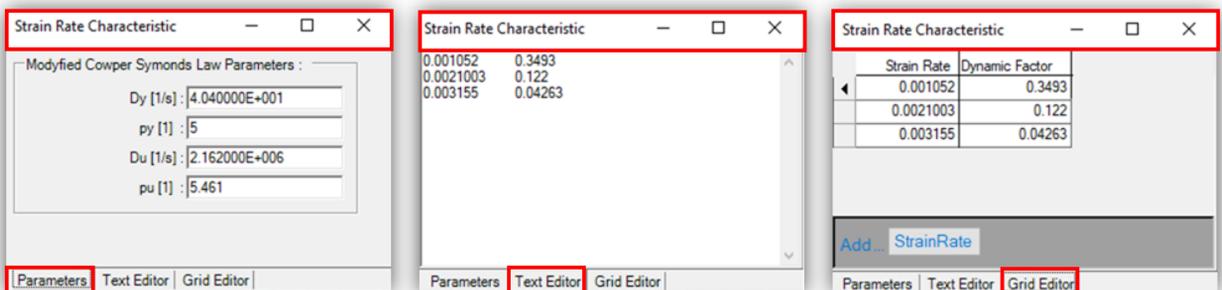
## MODIFIED COWPER SYMONDS 2

The only difference between Modified Cowper Symonds 1 and Modified Cowper Symonds 2 types lies in the definition of coupling function that now is specified as an array of strain strain-rate pairs.

$$\frac{\sigma(\epsilon, \dot{\epsilon})}{\sigma^s(\epsilon)} = [D_y(\dot{\epsilon}) - D_u(\dot{\epsilon})] \text{Array}(\epsilon, \dot{\epsilon}) + D_u(\dot{\epsilon})$$

The discrete definition of the coupling function provides the user with more flexibility in defining the dynamic response of material.

The parameters of the Modified Cowper Symonds 2 type are defined in a "triple-tabs" editor, containing: Parameters, Text Editor and Grid Editor tabs.



# STRAIN RATE CHARACTERISTIC

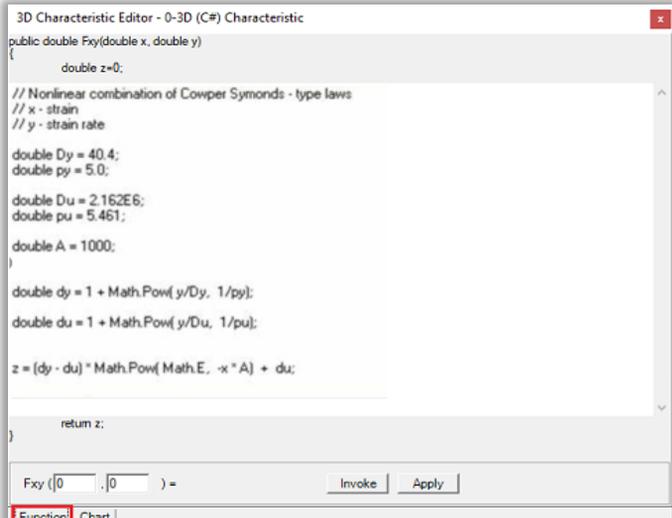
## THE USER DEFINED FUNCTION – 3D CHARACTERISTIC

Any programmable definition of the dynamic factor,  $D_x \rightarrow (\epsilon, \dot{\epsilon})$ , can be defined by the user by means of the 3D Characteristic type of the VCS.

The commands sequence: File → New → Characteristic 3D

creates an empty Characteristic on the Characteristics Branch of the Solution Explorer tree and opens the 3D Characteristic Editor window.

The body of the one-independent-variable function  $f(x,y)$  must be defined by the user in C# language. Note that the domain of the function (in this case maximal strain to fracture) is defined separately.



```
3D Characteristic Editor - 0-3D (C#) Characteristic
{
    public double Fxy(double x, double y)
    {
        double z=0;

        // Nonlinear combination of Cowper Symonds - type laws
        // x - strain
        // y - strain rate

        double Dy = 40.4;
        double py = 5.0;

        double Du = 2.162E6;
        double pu = 5.461;

        double A = 1000;
    }

    double dy = 1 + Math.Pow(y/Dy, 1/py);
    double du = 1 + Math.Pow(y/Du, 1/pu);

    z = (dy - du) * Math.Pow(Math.E, -x * A) + du;

    return z;
}
Fxy ([0] [0] [0]) =
[Function] [Chart]
```

The function takes two independent variables  $(x,y)$  and returns  $z$  as a function value. When the 3D Characteristic object is defined as a Strain Rate Characteristic type the  $x$  and  $y$  variables correspond to strain and strain rate, respectively, while  $z$  corresponds to resulting value of dynamic factor.

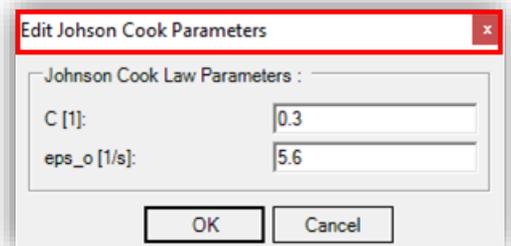
## JOHNSON COOK

Johnson Cook representation of the dynamic factor,  $D_x \rightarrow (\epsilon, \dot{\epsilon})$ , has the form of logarithmic function (the temperature factor is not defined in VCS applications):

$$(\sigma(\epsilon, \dot{\epsilon})) / (\sigma^A(\epsilon)) = 1 + C \cdot \ln(\dot{\epsilon} / \dot{\epsilon}_0)$$

After clicking on the '...' button in the Strain Rate Characteristic field in the Properties window the Edit Johnson Cook Parameters window will open.

In this window the two user-defined constants: strain-rate hardening constant  $C [1]$  and reference strain rate sensitivity  $\dot{\epsilon}_0 [1/s]$  are defined.



The Johnson Cook definition of the dynamic factor is not suitable for direct applications in numerical schemes. The ambiguity of this representation follows from the character of logarithmic function, which equals zero for argument equal unity and tends to negative infinity when argument approaches zero.



# STRAIN RATE CHARACTERISTIC

## JOHNSON COOK

In terms of material response, it means that material is not strain rate sensitive for strain rate ( $\dot{\epsilon} = \dot{\epsilon}_0$ ) and softens for smaller strain rates. For strain rates smaller than ( $\dot{\epsilon} = \dot{\epsilon}_0 / e^{(1/C)}$ ) Johnson Cook law predicts negative stresses and leads to instantaneous instability of the system due to artificial production of stress energy.

Therefore, in practical applications rate response of the material in this range must be approximated by means of other law. The appealing solution to this problem is a switch function that sets the dynamic factor to unity for strain rates smaller than  $\dot{\epsilon}_0$ .

However, this method is not suitable for dynamic calculations since it results in artificial impact-type material response when strain rate crosses the  $\dot{\epsilon}_0$  limit. In VCS the ambiguous region of strain rate response,  $0 \leq (\dot{\epsilon} \leq 4 \cdot \dot{\epsilon}_0)$ ; is approximated by power function that smoothly connects with logarithmic Johnson Cook curve.

# HARDENING FACTOR

Hardening factor,

$$\eta \quad (-1 \leq \eta \leq 1),$$

is a material constant, which describes hardening of material on complex straining paths and is compatible with the energy equivalent stress measure utilized in the macro element concept. The deformation mechanisms of the Super Folding Elements involve several loading-unloading-reloading deformation paths. For example, the rolling deformations present on the traveling hinge lines involve closed-strain-path loops on which material is bend and then re-bend to the initial state of strain.

Another example is the compression of material on the toroidal surface 1 followed by tensile straining on the conical surface 4.

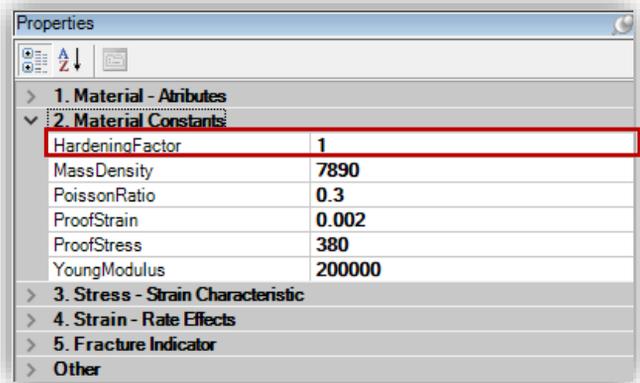
The energy absorption on such complex paths depends on the type of material hardening.

It is worth noting that density of dissipation in the case of pure kinematic hardening of material simply equals:

$$e = \sigma_y (|\epsilon_c| + |\epsilon_t|),$$

where  $\epsilon_c$  and  $\epsilon_t$  correspond, respectively, to maximal tensile and compressive strains ( $|\epsilon_c| = |\epsilon_t|$  on the closed loop strain path)

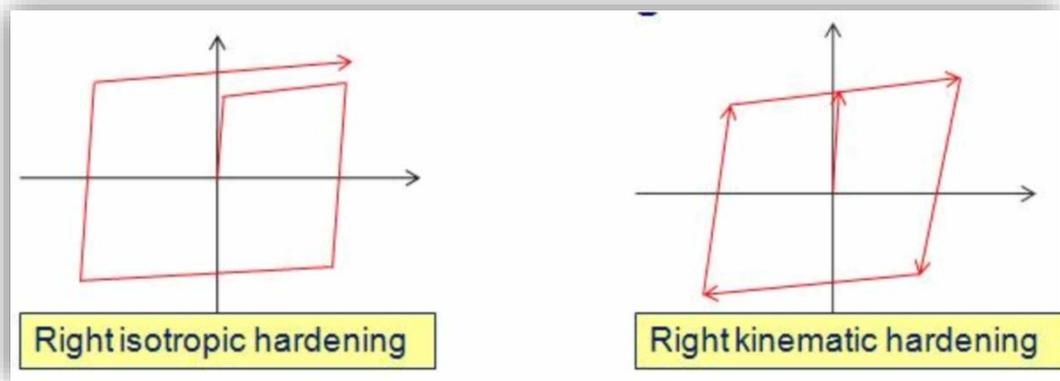
The hardening factor equal to unity,  $\eta = 1$ , corresponds to pure isotropic hardening, while  $\eta = 0$  corresponds to pure kinematic hardening.



Properties	
1. Material - Attributes	
2. Material Constants	
HardeningFactor	1
MassDensity	7890
PoissonRatio	0.3
ProofStrain	0.002
ProofStress	380
YoungModulus	200000
3. Stress - Strain Characteristic	
4. Strain - Rate Effects	
5. Fracture Indicator	
Other	



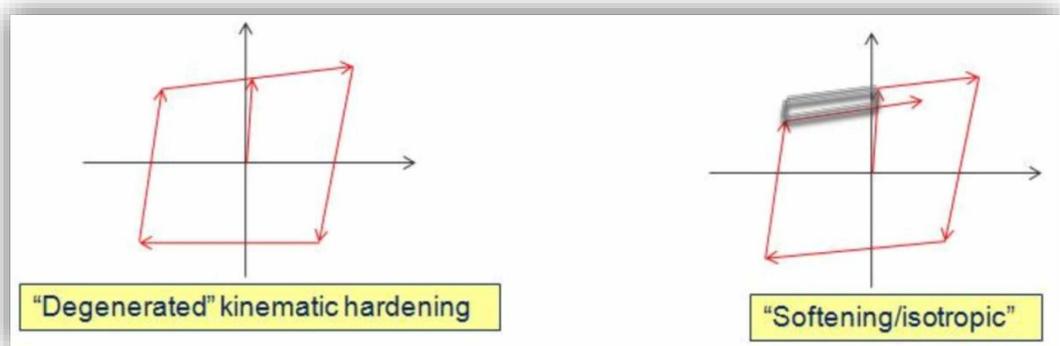
# HARDENING FACTOR



*Schematic illustration of loading-unloading-reloading path, respectively, for classic isotropic and kinematic hardening laws.*

*In real-world materials, especially new grades of HSS and UHSS, the shape of the stress-strain characteristic on re-loading paths differs significantly from that observed on the primary loading path. Schematic illustration of experimentally observed stress-strain loops is illustrated in Figure 45. For example, the shadowed area on the last graph in figure below illustrates the "energy deficiency" with respect to classic kinematic hardening law. Evidently, in this case the energy absorption density is smaller than lowest possible dissipation defined in classic theory of plasticity (kinematic hardening).*

*In VCS the discussed phenomenon is accounted for by negative values of the hardening factor. For negative  $\eta$  the energy density predicted by classic kinematic hardening law is reduced by the factor  $1+\eta$  (eta negative).*



*Schematic illustration of stress-strain variation for selected grades of HSS and UHSS materials (closed strain loops). The shaded area illustrates the "deficiency of energy density" with respect to classic kinematic hardening law.*

*Note that the hardening factor cannot be defined theoretically and/or numerically. It must be defined on basis of carefully conducted experiments on crushed prismatic members.*

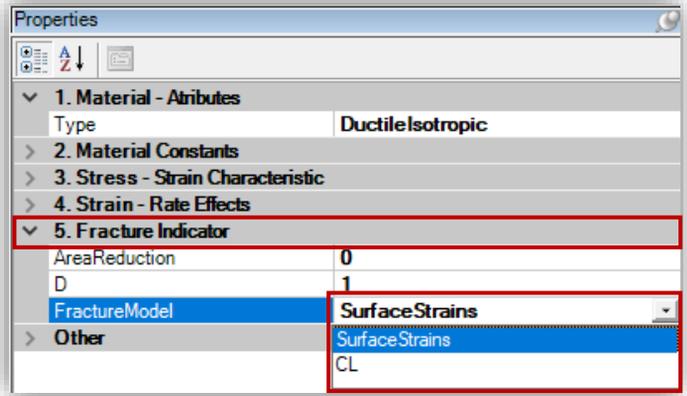


# FRACTURE INDICATOR

VCS implements two types of ductile fracture criteria:

- Surface strains
- Cockcroft-Latham/Norris.

In the Fracture Model field in the Fracture Indicator section of the Properties window you can find the drop-down list containing those two types of ductile fracture criteria

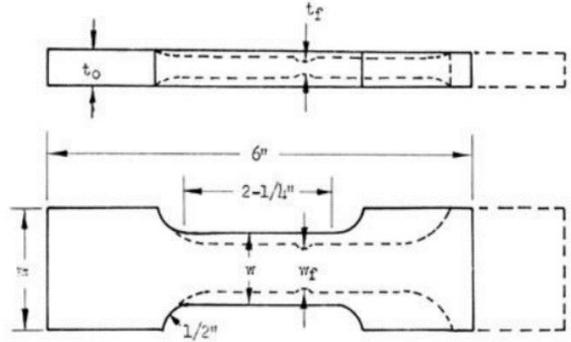


## SURFACE STRAINS

The Surface Strains fracture criterion was developed by Datsko and Yang. This criterion correlates neck area reduction in standard tensile test on sheet metal with critical radius of bending deformations.

Crucial for a successful fracture prediction by means of this criterion is careful definition of the Area Reduction factor  $A_r$  [%]. Area reduction is measured from the deformed tensile specimens after the tensile test.

The cross-section area of the tensile specimen before the test is  $A_o = w \cdot t_o$ . After the test the area is measured in the thinnest cross section and equals  $A_f = w_f \cdot t_f$



The non-dimensional Area Reduction factor calculated as:

$$A_r = \frac{A_o - A_f}{A_o} \cdot 100 \text{ [%]}$$

should be specified as an input parameter in the Area Reduction field of the Properties window.

The Surface Strains criteria uses a non-dimensional scalar parameter,  $D$ , - referred to as the Fracture Indicator - as a measure of ductile fracture.

Damage function is defined as path integral, where  $\sigma_{ef}$  - effective stress and  $\sigma_h$  - hydrostatic stress:

$$\bar{D}(\varepsilon_{eq}) = \int_{0 \rightarrow \varepsilon_{eq}} f\left(\frac{\sigma_h}{\sigma_{ef}}\right) d\varepsilon_{eq}$$

When fracture is initiated the damage function reaches its critical value  $\bar{D}(\varepsilon_{eq}) = \bar{D}_{cr}$   
The fracture indicator,  $D$ , is defined as a scalar measure,  $0 \leq D \leq 1$ , that indicates how close to the fracture initiation is material at given level of straining



# FRACTURE INDICATOR

$$D = \frac{\bar{D}(\varepsilon_{eq})}{\bar{D}_{cr}}$$

Fracture occurs when  $D = 1$  according to the respective theory. This value of can be adjusted by the user by specifying  $D$  in the range  $0 \leq D \leq 1$

This decreases the equivalent strain that marks the point of ductile fracture initiation.

## COCKCROFT-LATHAM/NORRIS (CL)

The Cockcroft-Latham/Norris criterion covers a wide range of stress triaxiality factor variation and provides a good approximation for fracture indicator in the case of arbitrary stress state.

The triaxiality parameter,  $\sigma_h / \sigma_{ef}$ , is an independent variable of the damage function. In the case of proportional monotonic loading in the strain space, defined by the constant ratio of two principal strain rates,  $\alpha = \dot{\varepsilon}_1 / \dot{\varepsilon}_2$ , this parameter can be defined analytically as shown e.g. by Wierzbicki:

$$\frac{\sigma_h}{\sigma_{ef}} = \frac{1}{\sqrt{3}} \frac{1 + \alpha}{\sqrt{1 + \alpha + \alpha^2}}$$

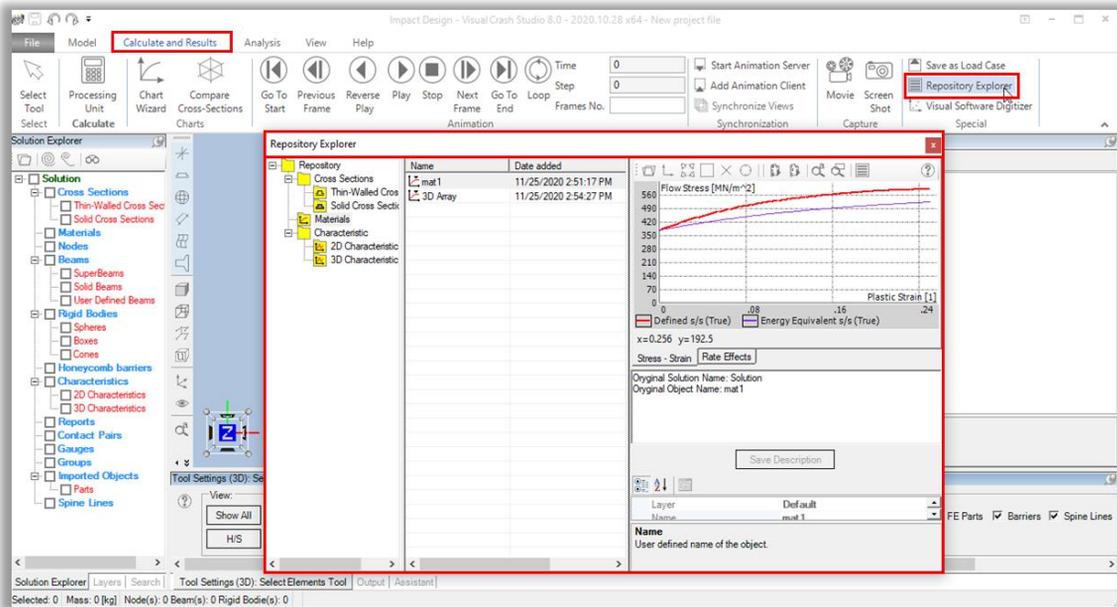
The Cockcroft-Latham/Norris criteria uses a non-dimensional scalar parameter,  $D$ , - referred to as the Fracture Indicator - as a measure of ductile fracture. Fracture occurs when  $D = 1$  according to the respective theory. This value of can be adjusted by the user by specifying  $D$  in the range  $0 \leq D \leq 1$

This decreases the equivalent strain that marks the point of ductile fracture initiation.



# THE MATERIAL AND CHARACTERISTIC REPOSITORY

The commands sequence: Calculation and Results → Special section → Repository Explorer  
After clicking on 'Repository Explorer' button, the Repository window will appear.



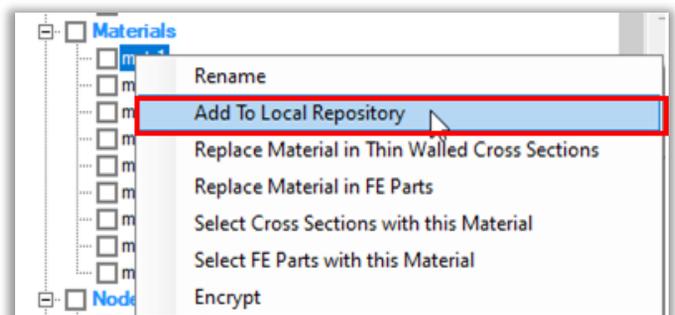
The Repository Explorer enables the user to create a personal Material and Characteristic library, which can be opened and used whenever working on a new solution. You may now save any Material or Characteristic and easily use it in a new Solution. You may also export any Material or Characteristic from a Solution into the Repository.

You may organize saved Materials and Characteristics by ascribing them to a selected folder (listed on the left side of the "Repository Explorer" window). You may also create new folders, transfer or copy objects from one folder to another and so on. This will enable you to create a personal, easy to use library of Materials and Characteristics. In order to create or delete a folder right-click on one of the folders on the left column of Repository Explorer. You may then select the "Add folder" tool to create a new folder or "Delete folder" tool. To move folders or object to new location simply drag and drop it. Remember that similarly to solution explorer in VCS you can't move material into characteristics folder etc.

In the middle of the "Repository Explorer" window you will find a list of Materials or Characteristics together with the date of its addition to the repository and the date of the latest modification. You may sort the Materials and Characteristics by name (alphabetically) by clicking on the "name" button, by the date of addition (click on "Date added") or by the date of last modification (click on "Date modified").

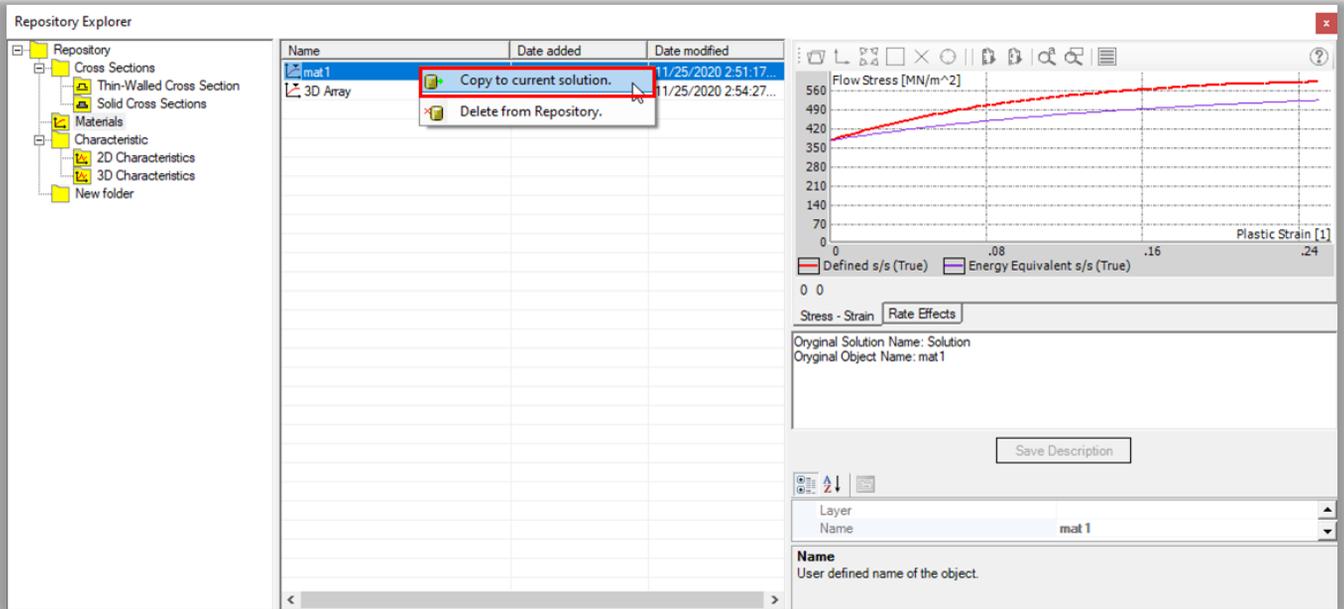
On the right side of the 'repository Explorer' window you will find the detailed 'read - only' properties of the selected Material or Characteristic.

In order to copy a Material (or a Characteristic) from the Solution into the Repository, left-click on the chosen Material in the Solution Explorer window, and then right-click. After choosing the "Add to Local Repository" a window will appear, in which you may describe the selected Material. The described Material will be automatically added to the main material folder in the Repository.



# THE MATERIAL AND CHARACTERISTIC REPOSITORY

In order to import a Material (or Characteristic) from the Repository into the Solution you need to right-click on the required Material and choose the "Copy to current solution" option. The Material will be automatically added to the Solution Explorer window.



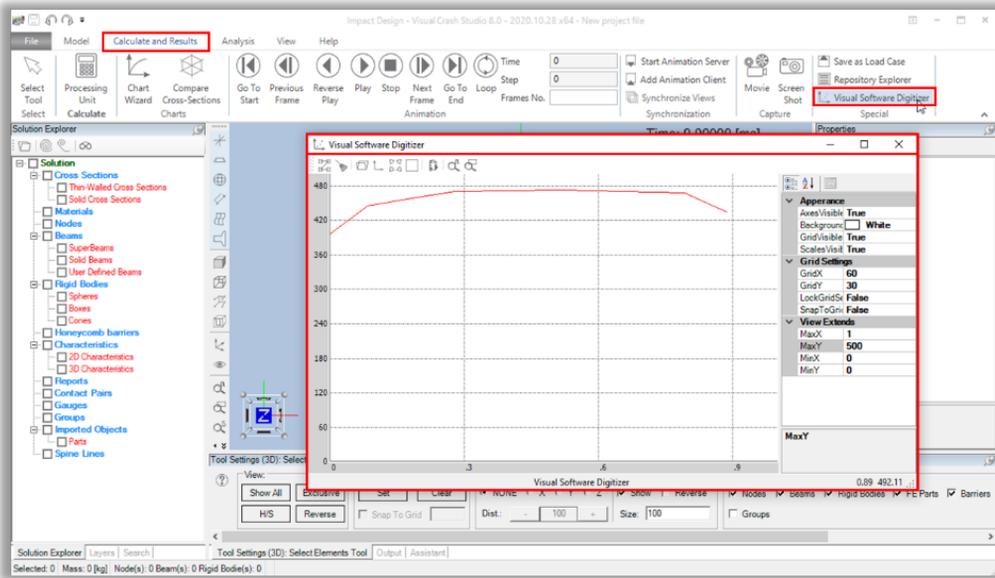
All the data gathered in the Repository Explorer will be automatically saved on your computer's drive in a separate folder. In order to find the Repository data, open the "Documents" folder, then find and open the "Visual Crash Studio" folder in which you will find the "Repository" folder with "repository.rvcs" file in it. You may easily copy the "repository.rvcs" file creating a backup of all of the Materials and Characteristics gathered in the Repository Explorer.



# VISUAL SOFTWARE DIGITIZER

The Visual Software Digitizer (VSD) enables the user to import a picture representing any chart or graph of a function and easily translate it into numerical data, which can be later used to define the properties of a new VCS Material or VCS Characteristic.

The commands sequence: Calculation and Results → Special section → Repository Explorer



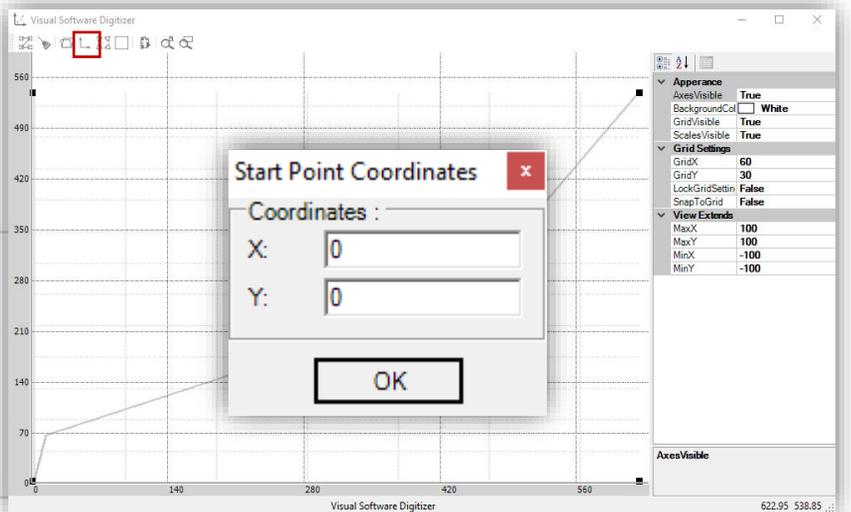
Visual Software Digitizer

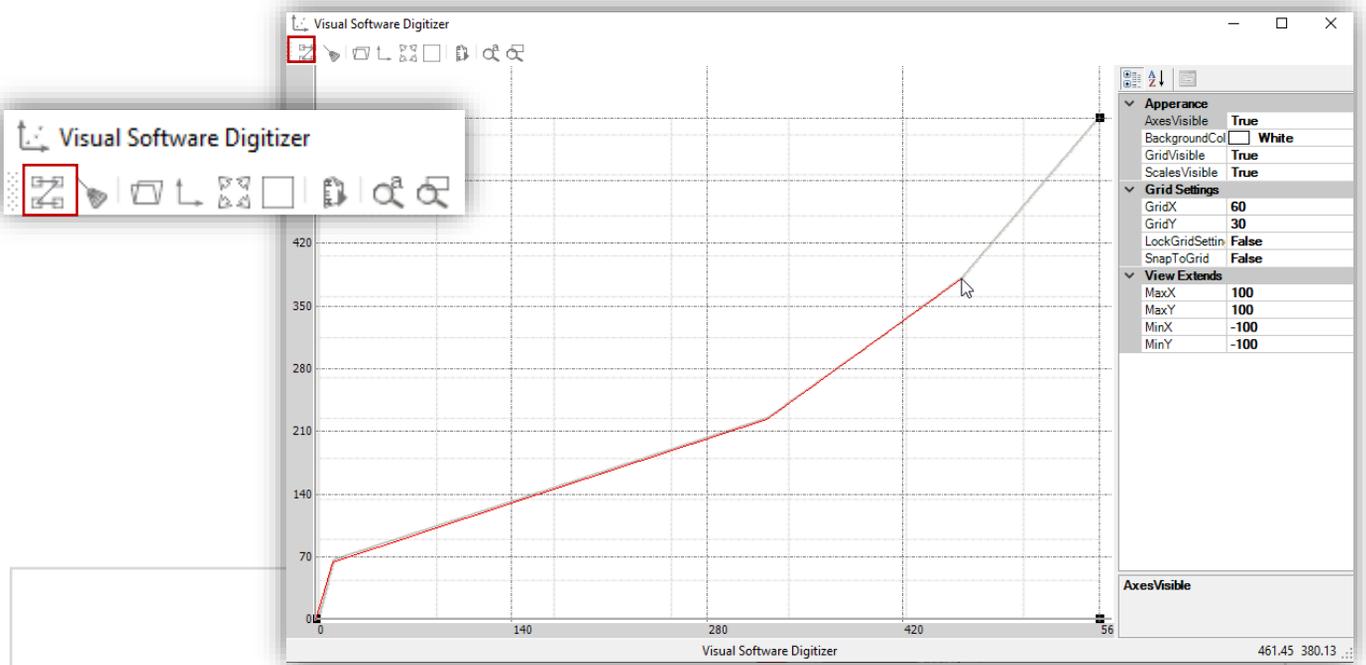
## Visual Software Digitizer tool

Visual Software Digitizer

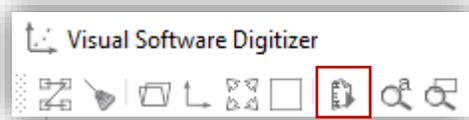
In the Visual Software Digitizer window click on the "Open" button. Now you may import any picture (for example a bitmap or a scan of a document) into the Visual Software Digitizer.

The imported picture will be visible in the main window of the VSD. Use the "Define Axis", "Size Background" and / or "Pan Background" tools to manage the imported picture



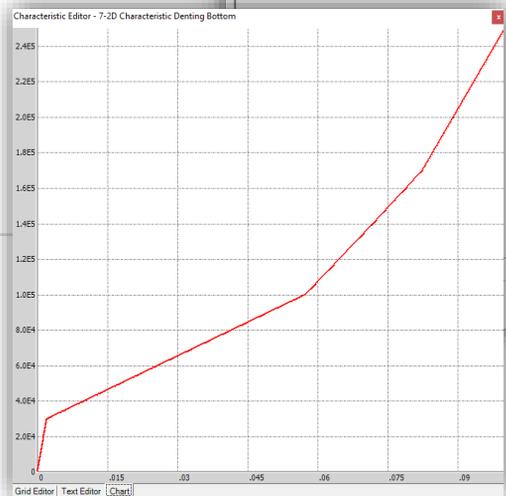
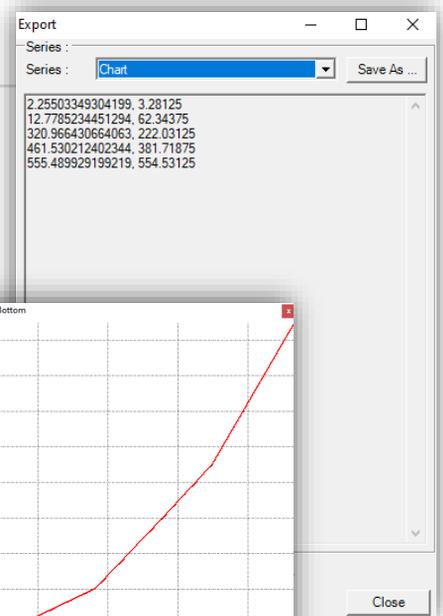


After clicking on the "Add Point" tool you can enter points on the imported graph of a function. A red line will be automatically drawn between every two points. The coordinates of each point will be saved, in the correct order based on horizontal axis coordinates.



Click on the "Export" icon to export the text list of points entered in the picture view. This data can be later used to define parameters of a new VCS Characteristic or Material.

In case of a new VCS material, you can use the exported text file or clipboard data to create the Stress Strain Characteristic of the Material or any other characteristics.

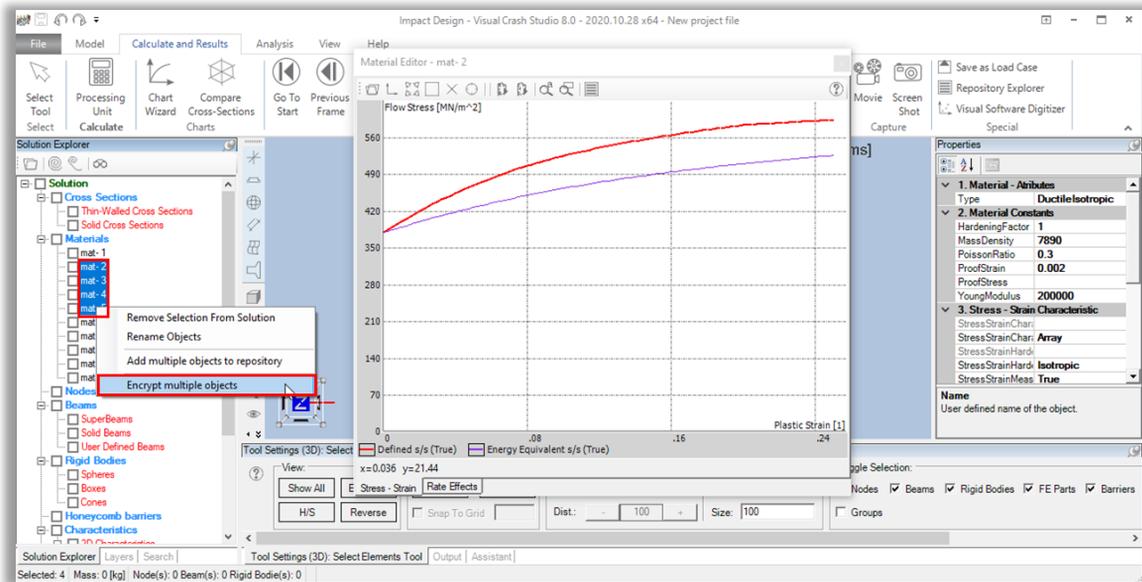


(\* ) please note that points' coordinates are automatically exported with a coma separating the two numbers. Make sure to remove the coma before entering point's coordinates to the VCS Characteristic Editor or for the purpose of material definition.



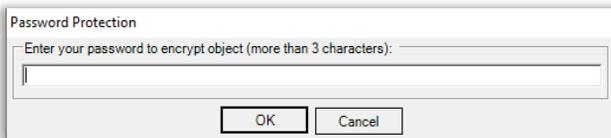
# MATERIAL AND CHARACTERISTIC ENCRYPTION

This feature enables the User to hide all properties of a Material or Characteristic and at the same time enable its application by other VCS users.



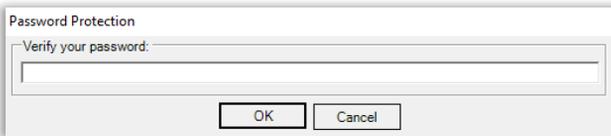
After selecting a standard Material in the Solution Explorer window, you can see and edit its properties in the Properties window. Also, the Material Editor window will appear on the screen in which Stress-Strain and Rate Effect charts can be seen.

If you wish to code all those information, right click on a Material (or Characteristic) and select the "Encrypt" option. There is also possibility to encrypt the multiple objects at once, by selecting objects with Ctrl on keyboard and mouse right click. This functionality enables you to secure all Material data by a personal password. At the same time the encrypted Material can be used in any future VCS Solutions.



After selecting the "Encrypt" option a "Password Protection" window will appear on the screen.

Please remember to choose your password carefully because there is no possibility of recovering it in the future!



After the password is entered the software will ask for its verification, as illustrated in the picture on the left.

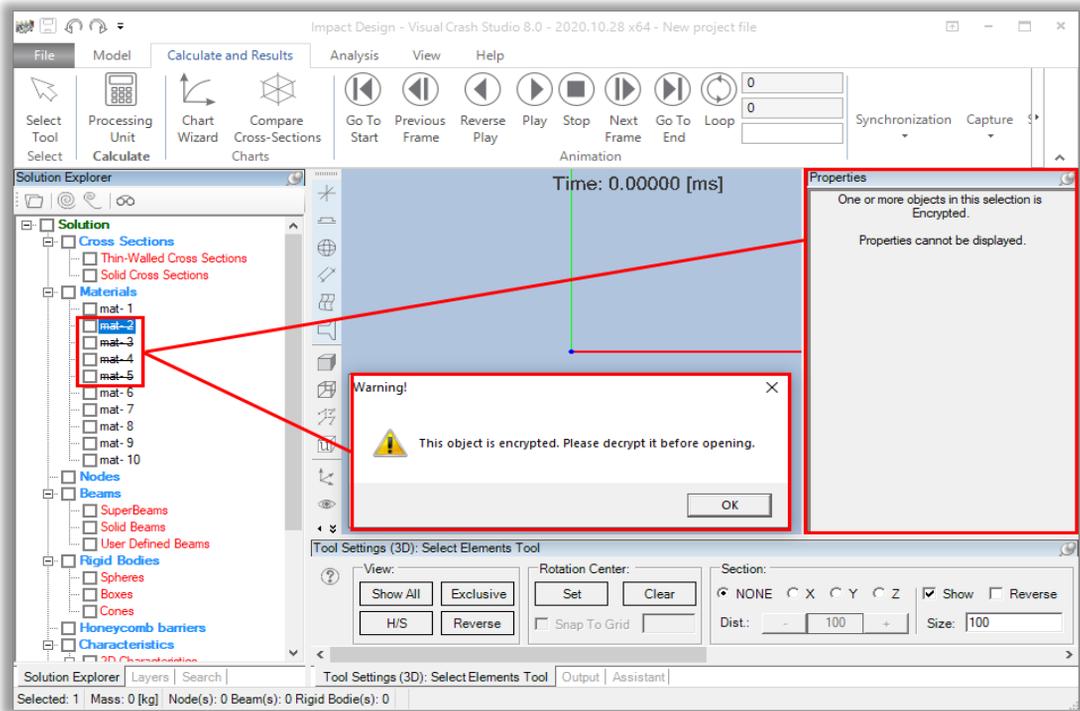
Encrypted materials can be recognized by the strikethrough on their names.

If an encrypted Material or Characteristic is selected, the message "This Object is encrypted. Please decrypt it before opening" will appear instead of the Material or Characteristic Editor window. The Material or Characteristic properties will be unavailable and cannot be viewed or edited.

Nevertheless, an encrypted Material or Characteristic can be used in a VCS Solution and will not have negative impact on the calculation process.

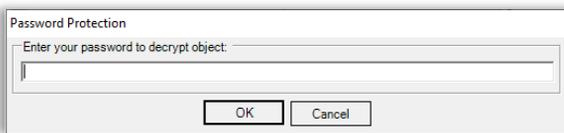
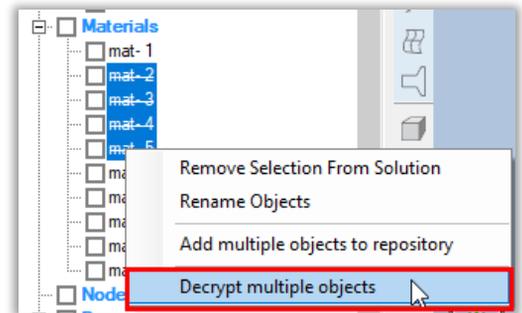


# MATERIAL AND CHARACTERISTIC ENCRYPTION



The properties and Material data can be viewed only after decrypting them with the usage of the previously created password.

In order to decrypt a Material or Characteristic right click on it, in the Solution Explorer window and select the "Decrypt" option – as shown in the picture on the right. The "Password Protection" window will appear on the screen.



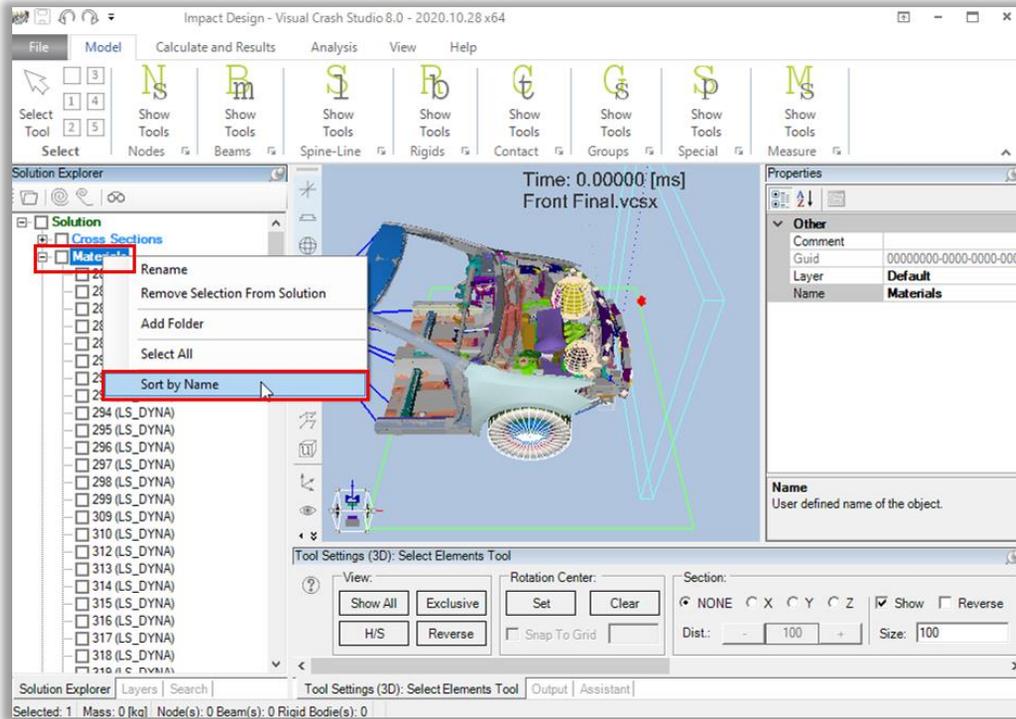
After entering the correct password in it, all information will be decoded.

Material or Characteristic properties will be fully available for edition in the properties window.



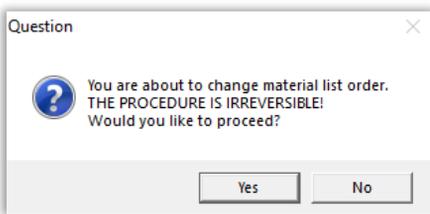
# *SORT MATERIALS BY NAME*

*This feature enables the User to sort all Materials in the Solution by Name.*



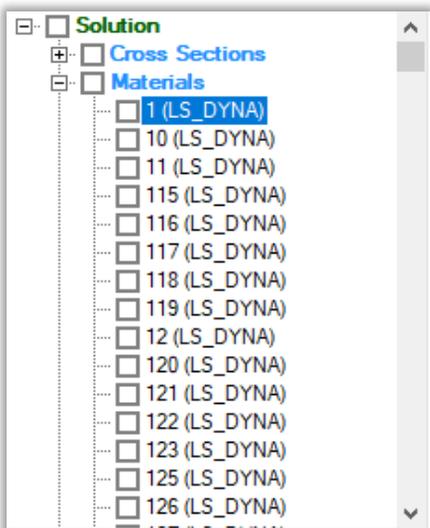
*The default order of displaying materials in the Solution Explorer Tree, depends on the order in which the materials are added to the Solution.*

*To sort materials, select Material section in the Solution Explorer window and do the right button click on it. From drop down list select Sort by Name option.*



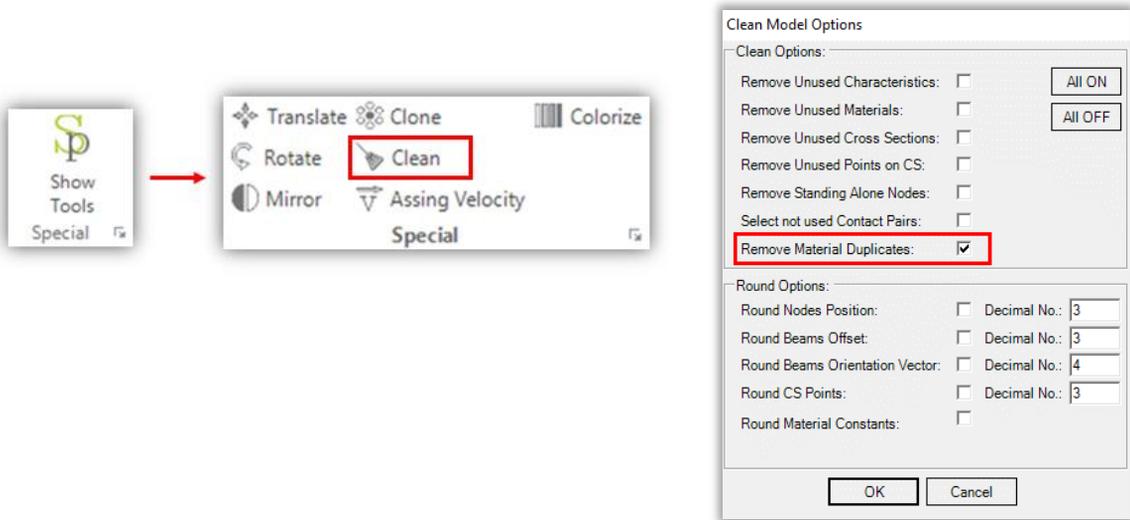
*To finish the procedure, confirm by selecting Yes button in the Question window. Note that changes are irreversible.*

*Materials are sorted in alphabetical order, starting with numbers*

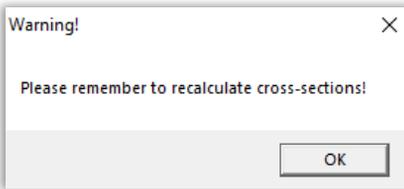


# REMOVE DUPLICATED MATERIALS

This feature enables to remove all duplicated materials in the Solution that have the same material properties even when they have different names.

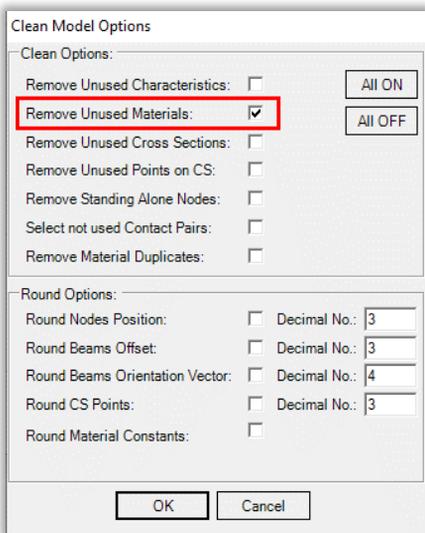


To remove duplicated materials, select Clean tool from Special section in Main Toolbar. In the Clean Model Options window select Remove Material Duplicates checkbox and confirm by OK button. It is recommended to remove duplicated materials directly after importing materials.



Duplicated materials used in Cross Section will be replaced by only one representative material model from group of duplicated materials. In that case it is required to recalculate cross sections that contain replaced materials.

# REMOVE UNUSED MATERIALS



To reduce the size of the file and keep the model clean, all unused materials can be removed from the Solution just by one click.

In Clean tool select the Remove Unused Materials checkbox and confirm by Ok button, to remove all unused materials.

