METALLIC PLASTICITY MODELLING – ABAQUS FEM CODE

Compiled by:	Álvaro José Martínez	27/07/2022
Supervised by:	Álvaro José Martínez	27/07/2022
Approved by:	Álvaro José Martínez	27/07/2022

List of Document Changes

Issue	Date	Remarks
1	20/01/2015	First Issue.
2	18/03/2016	K formula (page 13) has been corrected
3	27/07/2022	3.3.5. Ramberg-Osgood Idealization – Sources of error

CONTENTS

REFERENCES **5**
1. INTRODUCTION **6**
2. MATERIAL STRENGTH PARAMETERS **7**
2.1. YOUNG'S MODULUS, E 8
2.2. YIELD STRENGTH - F_{TY} 8
2.3. ULTIMATE TENSILE STRENGTH - F_{TU} 8
2.4. PERCENT ELONGATION - % E 8
3. MATERIAL MODELLING IN ABAQUS **9**
3.1. TRUE STRESS AND LOGARITHMIC STRAIN (TRUE STRAIN) 9
3.2. PLASTIC ENTRIES IN ABAQUS CODE 9
3.3. IDEALIZATION OF THE STRESS-STRAIN CURVE 10
3.3.1. Elastic-Plastic Idealization 10
3.3.2. Elastic-Linear strain hardening Idealization 11
3.3.3. Ramberg-Osgood Idealization 12
3.3.4. Ramberg-Osgood Idealization for ABAQUS Code 13
3.3.5. Ramberg-Osgood Idealization – Sources of error 18
4. DEFINING MATERIAL DATA IN ABAQUS ENVIROMENT **19**
5. DFEM MODELLING GUIDELINES FOR NON-LINEAR ANALYSIS WITH ABAQUS (PLASTICITY) **20**
5.1. SHELL AND SOLID ELEMENT TYPES 20
5.1.1. BENDING DOMINANT PROBLEMS 20
5.1.2. CONTACT DOMINANT PROBLEMS WITHOUT BENDING 20
5.1.3. CONTACT DOMINANT PROBLEMS WITH BENDING 20
5.2. MESH DENSITY 20
5.2.1. STRESS DISCONTINUITY 20
5.2.2. FILLET RADIUS 20
6. POST-PROCESSING OF NON-LINEAR PLASTICITY ANALYSIS **22**
6.1. RESERVE FACTOR 22
6.2. PLASTIC ONSET FOR RAMBERG – OSGOOD IDEALIZATION 22
7. EXAMPLE **23**
7.1. GEOMETRY 23
7.2. MESH 23
7.3. MATERIAL 24
7.4. LOAD 24
7.5. BOUNDARY CONDITIONS 24
7.6. RESULTS 25

7.6.1. MODEL 01 – Material Idealization: Elastic-Linear strain hardening 25
7.6.2. MODEL 02 – Material Idealization: Ramberg – Osgood 25
7.6.3. Neuber Plasticity Correction 25
7.6.4. Summary 25

REFERENCES

The following references have been used:

ID	TITLE	REFERENCE	ISSUE	DATE	SOURCE
Ref. 1.	ABAQUS 6.12 Manual		6.12	2012	SIMULIA
Ref. 2.	A350 Detailed FEM Modelling guidelines for Linear and Non Linear Analysis	V53PR0906133	3.0	2011	AIRBUS
Ref. 3.	Guía para la realización de Análisis de Estructuras mediante Modelos de Elementos Finitos de Detalle	NT-T-ADP-10003	A	2010	AIRBUS MILITARY
Ref. 4.	Description Stress-Strain curves by Three Parameters	---	---	1943	Ramberg – Osgood
Ref. 5.	MMPDS Metallic Material Properties Development and Standardization	MMPDS-06		2011	FAA

1. INTRODUCTION

This document presents the different ways to model the plasticity behavior in ductile metals and its implementation in the FEM ABAQUS Code.

All concepts are based in the engineering tension test, which is widely used to provide basic design information on the strength of materials and as an acceptance test for the specification of materials.

In the tension test a specimen is subjected to a continually increasing uniaxial tensile force while simultaneous observations are made of the elongation of the specimen. The parameters, which are used to describe the stress-strain curve of a metal, are the tensile strength, yield strength or yield point, percent elongation, and reduction of area. The first two are strength parameters; the last two indicate ductility.

A typical response is shown in following figure:

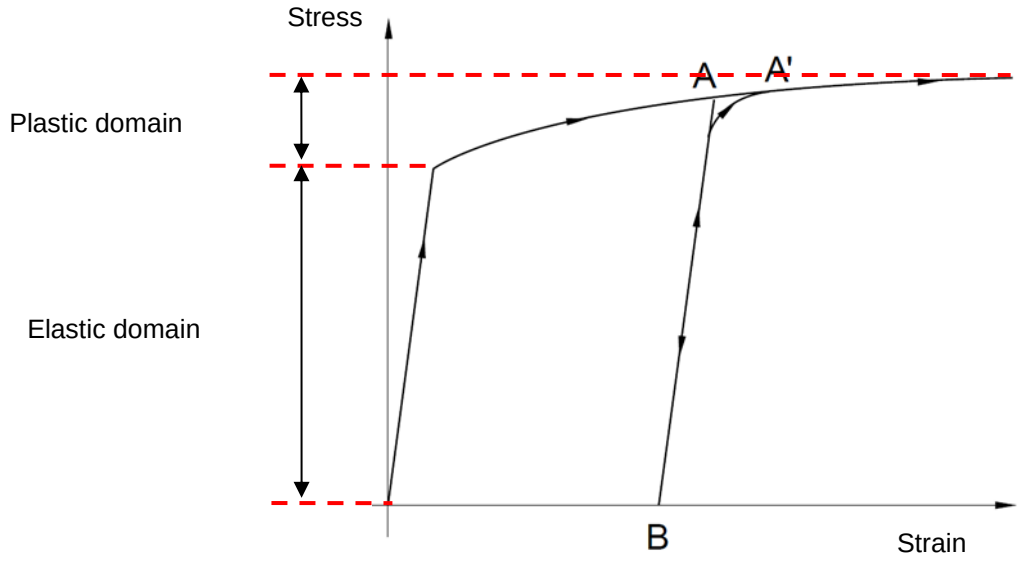


Figure 1-1 Typical response of uni-axial test for a ductile metal

2. MATERIAL STRENGTH PARAMETERS

The uniaxial tensile test is a common standard test and is a valuable method of determining important mechanical properties of engineering materials.

Following figure shows the stress-strain relationship obtained from a uniaxial tensile test:

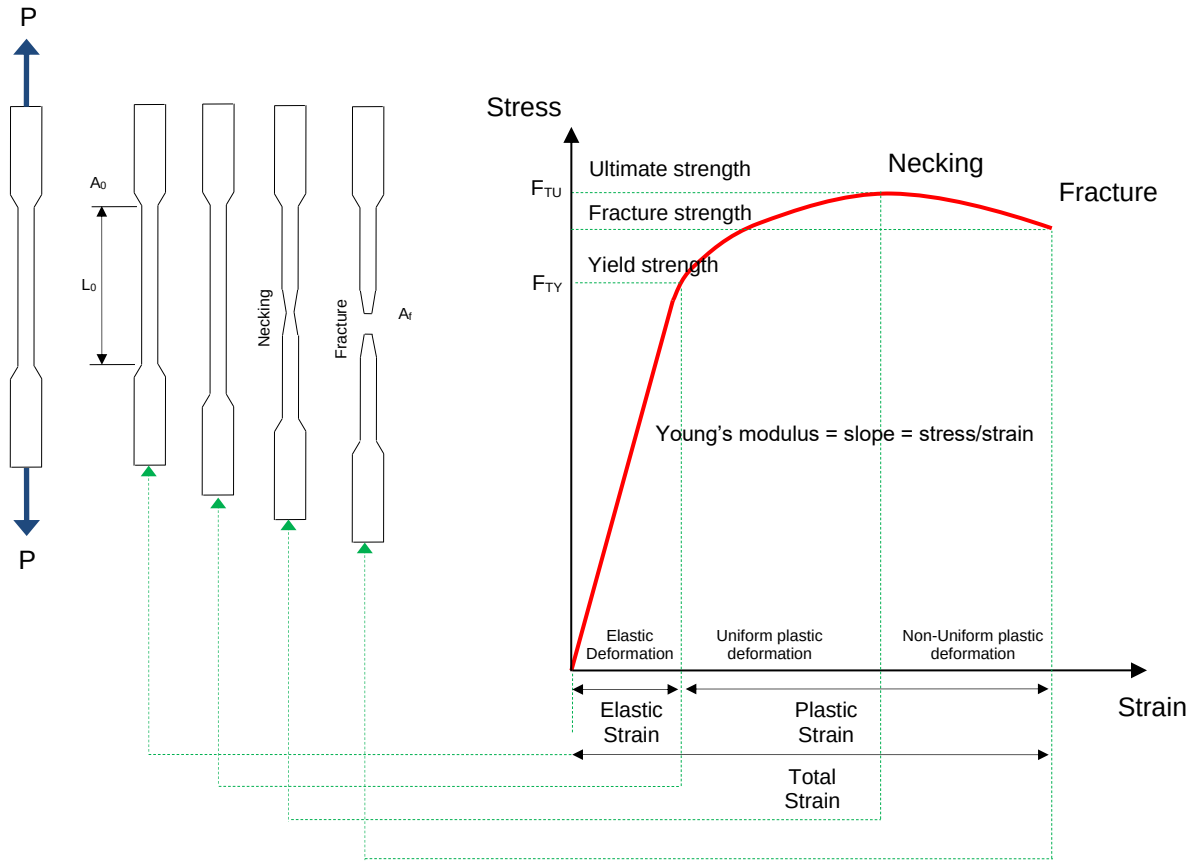


Figure 2-1 Stress-strain relationship under uniaxial tensile loading

Following chapters explain how to obtain the material strength parameters from test.

2.1. YOUNG'S MODULUS, E

During elastic deformation, the engineering stress-strain relationship follows the Hooke's Law and the slope of the curve indicates the Young's modulus (E)

$$E = \frac{\sigma}{\varepsilon}$$

2.2. YIELD STRENGTH - F_{TY}

The stress at which a material exhibits a specified permanent deformation.

This stress is usually determined by the offset method, where the strain departs from the linear portion of the actual stress–strain diagram by an offset unit strain of 0.002.

$$F_{\mathrm{TY}} = \frac{P_{(\mathrm{strain_offset=0002})}}{A_0}$$

Where:

- A_0 = Original cross sectional area
- $P_{(strain_offset=0.002)}$ = Load at 0.2% strain

2.3. ULTIMATE TENSILE STRENGTH - F_{TU}

The ultimate strength of a material in uniaxial tension is the maximum tensile stress that the material can sustain calculated on the basis of the greatest load achieved prior to fracture.

$$F_{\mathrm{TU}} = \frac{P_{\mathrm{MAX}}}{A_0}$$

Where:

- A_0 = Original cross sectional area
- P_{MAX} = Maximum load

2.4. PERCENT ELONGATION - % E

Percent elongation is the increase in gage length, measured after fracture of the tensile specimen within the gage length, expressed as a percentage of the original gage length:

$$\%\ \mathrm{e} = \frac{L_f\text{-}\mathrm{L}_0}{L_0} \cdot 100$$

Where:

- L_0 = Initial gage length
- L_f = Length of the gage section at fracture

3. MATERIAL MODELLING IN ABAQUS

Ductile metallic materials are usually modeled as plastic behavior with isotropic hardening (the yield surface changes size uniformly in all directions such that the yield stress increases or decreases in all stress directions as plastic straining occurs).

Yield data should always be given in ABAQUS as ***TRUE STRESS VERSUS LOGARITHMIC STRAIN.***

3.1. TRUE STRESS AND LOGARITHMIC STRAIN (TRUE STRAIN)

Logarithmic strain (or true strain), ε, is defined as:

$$\varepsilon = \int_{L_0}^{L} \frac{\mathrm{dL}}{L} = \ln\left(\frac{L}{L_0}\right) = \ln(1+\varepsilon^{\mathrm{nom}})$$
$$\varepsilon^{nom} = \frac{\mathrm{L\text{-}L_0}}{L_0}$$

Where:

- L = Current gage length
- L_0 = Original gage length
- ε^{nom} = engineering or nominal strain

There are two ways to define stress:

- Engineering or nominal stress, σ^{nom}: the force at any time during the test divided by the initPLASial area of the test piece A_0; $\sigma^{nom} = F/A_0$
- True stress, σ: The force at any time divided by the instantaneous area of the test piece; $\sigma = F/A_i$, A_i is the instantaneous cross section of a test piece

Assuming that the piece volume is constant during deformation, A_i can be defined as a function of A_0 and ε^{nom}:

$$A_i L_i = A_0 L_0$$
$$\frac{A_0}{A_i} - 1 = \frac{L_i}{L_0} - 1 = \frac{L_i - L_0}{L_0} = \varepsilon^{\mathrm{nom}}$$
$$A_i = \frac{A_0}{1+\varepsilon^{\mathrm{nom}}}$$

Therefore, the true stress is defined as:

$$\sigma = \frac{F}{A_i} = \frac{F}{A_0}(1+\varepsilon^{\mathrm{nom}}) = \sigma^{nom}(1+\varepsilon^{\mathrm{nom}})$$

3.2. PLASTIC ENTRIES IN ABAQUS CODE

Material entries for plastic behavior with isotropic hardening in ABAQUS code are defined as following:

```
**************************************
** Elastic behavior
**************************************
*Material, name=material_name
*Elastic
young's module, poisson coefficient
**************************************
** Plastic behavior
**************************************
*Plastic
true_stress_1, logarithmic_plastic_strain_1
```

```
true_stress_2, logarithmic_plastic_strain_2
true_stress_3, logarithmic_plastic_strain_3
...
```

Where the logarithmic plastic strain is defined as the total logarithmic strain minus the elastic logarithmic strain:

$$\varepsilon_{ln}^{pl} \ln(1+\varepsilon^{\text{nom}}) \frac{\sigma}{E} \ln(1+\varepsilon^{\text{nom}}) \frac{\sigma^{nom}(1+\varepsilon^{\text{nom}})}{E}$$

3.3. IDEALIZATION OF THE STRESS-STRAIN CURVE

Because of the complex nature of the stress-strain curve, it has become customary to idealize this curve in various ways.

Following chapters shows typical idealizations for ductile metallic plastic behavior.

3.3.1. Elastic-Plastic Idealization

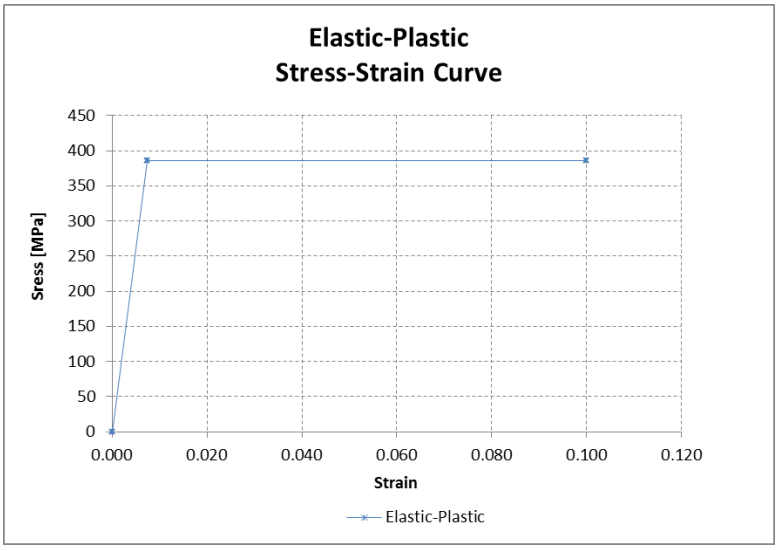


Figure 3-1 Elastic-plastic idealization

This stress-strain curve is defined through the following entries:

```
****************************************
** Elastic behavior
****************************************
*Material, name=material_name
*Elastic
E, ν
****************************************
** Plastic behavior
****************************************
*Plastic
```

$$F_{\text{ty}}\left(1+\frac{F_{\text{ty}}}{E}\right), 0.0$$

$$F_{\text{ty}}\left(1+\frac{F_{\text{ty}}}{E}\right), \ln(1+e)-\frac{F_{\text{ty,true}}}{E}$$

Where:

$$F_{\text{ty,true}} = F_{\text{ty}}\left(1+\frac{F_{\text{ty}}}{E}\right)$$

$$\text{e} = \text{elongation}$$

3.3.2. Elastic-Linear strain hardening Idealization

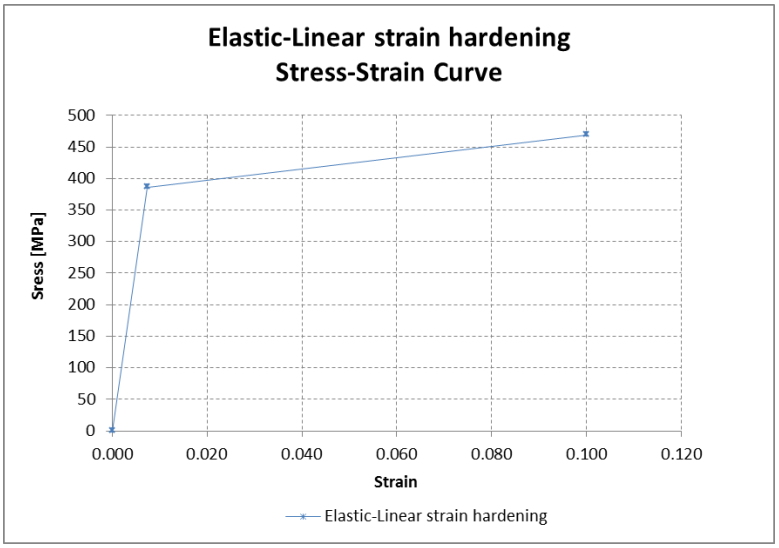


Figure 3-2 Elastic-plastic idealization

This stress-strain curve is defined through the following entries:

```
**************************************
** Elastic behavior
**************************************
*Material, name=material_name
*Elastic
E, ν
**************************************
** Plastic behavior
**************************************
*Plastic
```

$$F_{ty}\left(1+\frac{F_{ty}}{E}\right), 0.0$$

$$F_{tu,true}, \ln(1+e)-\frac{F_{ty,true}}{E}$$

Where:

$$F_{tu,true} = F_{tu}(1+e)$$

$$F_{ty,true} = F_{ty}\left(1+\frac{F_{ty}}{E}\right)$$

$$e = \text{elongation}$$

3.3.3. Ramberg-Osgood Idealization

Ramberg and Osgood proposed an analytical expression for the stress-strain curved in terms of three parameters, *E*, *K* and *n*:

$$\varepsilon = \varepsilon_{\text{elastic}} + \varepsilon_{\text{plastic}} = \frac{\sigma}{E} + K\left(\frac{\sigma}{E}\right)^n$$

Where *K* and *n* are material parameters which can be obtained taking only two points in the stress-strain curve (σ_1 ε_1 ; σ_2 ε_2)

$$\left.\begin{matrix}\sigma_1 = m_1 \cdot \mathrm{E} \cdot \varepsilon_1 \\ \sigma_2 = m_2 \cdot \mathrm{E} \cdot \varepsilon_2\end{matrix}\right\} \Rightarrow \begin{cases}\varepsilon_1 = \dfrac{\sigma_1}{m_1 \cdot \mathrm{E}} = \dfrac{\sigma_1}{E} + K\left(\dfrac{\sigma_1}{E}\right)^n \\ \varepsilon_2 = \dfrac{\sigma_2}{m_2 \cdot \mathrm{E}} = \dfrac{\sigma_2}{E} + K\left(\dfrac{\sigma_2}{E}\right)^n\end{cases}$$

Simplifying these equations can be obtained *n* and *K* for m_1 and m_2 (slopes of a line through the origin of points 1 and 2):

$$n = 1 + \frac{log\ \left(\frac{m_2}{m_1}\frac{1-m_1}{1-m_2}\right)}{log\ \left(\frac{\sigma_1}{\sigma_2}\right)}$$

$$K = \frac{1-m_1}{m_1}\left(\frac{\sigma_1}{E}\right)^{1-n}$$

Ramberg and Osgood defined that the best points to represent the stress-strain curve are:

$$\begin{matrix}m_1 = 0.7 \\ m_2 = 0.85\end{matrix} \Rightarrow \begin{matrix}\sigma_1 = \sigma_{0.7} \\ \sigma_2 = \sigma_{0.85}\end{matrix}$$

Therefore, n and K parameters and ε are defined as:

$$n = 1 + \frac{log\ \left(\frac{17}{7}\right)}{log\ \left(\frac{\sigma_{0.7}}{\sigma_{0.85}}\right)}$$

$$K = \frac{3}{7}\left(\frac{\sigma_1}{E}\right)^{1-n}$$

$$\varepsilon = \frac{\sigma}{E} + \frac{3}{7}\left(\frac{\sigma_{0.7}}{E}\right)\left(\frac{\sigma}{\sigma_{0.7}}\right)^n$$

Because $\sigma_{0.7}$ and $\sigma_{0.85}$ are not usually known in the material databases, the modified Ramberg-Osgood criterion is used, where the points F_y and F_u are chosen to define the Ramberg-Osgood parameters.

$$n = \frac{log\ \left(\frac{e - \frac{F_u}{E}}{0.002}\right)}{log\ \left(\frac{F_u}{F_y}\right)}$$

$$K = 0.002\left(\frac{E}{F_y}\right)^n$$

Where F_u is the ultimate strength, F_y is the yield strength and e the elongation.

Modified Ramberg-Osgood formulae for different stress behavior are defined as following:

Tension:

$$n_t = \frac{log\ \left(\frac{e - \frac{F_{tu}}{E_t}}{0.002}\right)}{log\ \left(\frac{F_{tu}}{F_{ty}}\right)}$$

$$\varepsilon = \frac{\sigma}{E} + 0.002\left(\frac{\sigma}{F_{ty}}\right)^{n_t}$$

Compression:

$$n_c = \frac{log\left(\frac{e-\frac{F_{tu}}{E_c}}{0.002}\right)}{log\left(\frac{F_{tu}}{F_{cy}}\right)}$$

$$\varepsilon = \frac{\sigma}{E} + 0.002\left(\frac{\sigma}{F_{cy}}\right)^{n_c}$$

Shear:

$$n_s = \frac{n_c + n_t}{2}$$

$$\varepsilon = \frac{\tau}{G} + 0.002\sqrt{3}\left(\frac{\tau}{F_{sy}}\right)^{n_s}$$

3.3.4. Ramberg-Osgood Idealization for ABAQUS Code

Material modelling in Abaqus FEM Code of Ramberg-Osgood idealization is shown through an example to facilitate its practical application.

- Material: Al 2024 T351 – B – LT – Range: [25.43 < t < 38.10]
- E = 73774 N/mm2 (Young Modulus)
- F_{tu}= 441.3 N/mm2 (Ultimate strength)
- F_{ty}= 303.4 N/mm2 (Yield strength)
- e = 7 % (strain at rupture)
- Off. Strain = 0.01 % (Plastic onset in Ramberg-Osgood curve)

Plastic material behavior in ABAQUS code is defined with entries *ELASTIC and *PLASTIC.

*ELASTIC entry defines the elastic response of the material. The elastic response is defined in terms of constant (with respect to strain) moduli, such as Young's modulus and Poisson's ratio for an isotropic material.

Because F_{ty} is defined for an offset unit strain of 0.002 in the stress–strain diagram, it is necessary to define plastic behavior in a previous stress point, σ_{ref}, in order to avoid discontinuities in plastic stress-strain diagram.

σ_{ref} is calculated for a fixed ratio (Offset Strain) between the elastic strain and the Ramberg-Osgood strain, in order to achieve the plastic onset in the Ramberg-Osgood curve. Following equation shows how to calculate the σ_{ref}:

$$\varepsilon - \frac{Off.Strain}{100} = \frac{\sigma_{ref}}{E}$$

$$\frac{\sigma_{ref}}{E} + 0.002\left(\frac{\sigma_{ref}}{F_{ty}}\right) - \frac{Off.Strain}{100} = \frac{\sigma_{ref}}{E}$$

$$\sigma_{ref} = F_{ty}\left(\frac{Off.Strain/100}{0.002}\right)^{\frac{1}{n}}$$

Offset Strain equal to 0.01 is recommended.

3.3.4.1. Ramberg-Osgood Coefficient

$$n_t = \frac{\log\left(\frac{e-\frac{F_{tu}}{E_t}}{0.002}\right)}{log\left(\frac{F_{tu}}{F_{ty}}\right)} = \frac{log\left(\frac{0.07-\frac{441.3}{73774}}{0.002}\right)}{log\left(\frac{441.3}{303.4}\right)} = 9.25$$

3.3.4.2. σ_{ref}

$$\sigma_{ref} = F_{ty}\left(\frac{Off.Strain/100}{0.002}\right)^{\frac{1}{n}} = 303.4\left(\frac{0.01/100}{0.002}\right)^{\frac{1}{9.25}} = 219.5MPa$$

3.3.4.3. Material Stress-Strain Curve (Engineering Curve)

$$\varepsilon^{nom} = \frac{\sigma^{nom}}{E} + 0.002\left(\frac{\sigma^{nom}}{F_{ty}}\right)^{n_t} = \frac{\sigma^{nom}}{73774} + 0.002\left(\frac{\sigma^{nom}}{303.4}\right)^{9.25}$$

σ^{nom} (MPa)	ε^{nom}
0.0	0.00000
109.7	0.00149
170.4	0.00232
207.0	0.00286
228.9	0.00325
219.5	0.00307
227.9	0.00323
236.3	0.00340
244.6	0.00359
253.0	0.00380
261.4	0.00405
269.8	0.00433
278.2	0.00467
286.6	0.00507
290.8	0.00529
295.0	0.00554
299.2	0.00581
303.4	0.00611
306.2	0.00632
308.9	0.00655
311.7	0.00679
314.4	0.00705
317.2	0.00732
319.9	0.00761
322.7	0.00791
325.5	0.00824
328.2	0.00859
331.0	0.00896
337.9	0.00999
344.8	0.01120
358.6	0.01424
372.4	0.01834
386.1	0.02385
399.9	0.03117
413.7	0.04085
427.5	0.05352
434.4	0.06123
441.3	0.07000

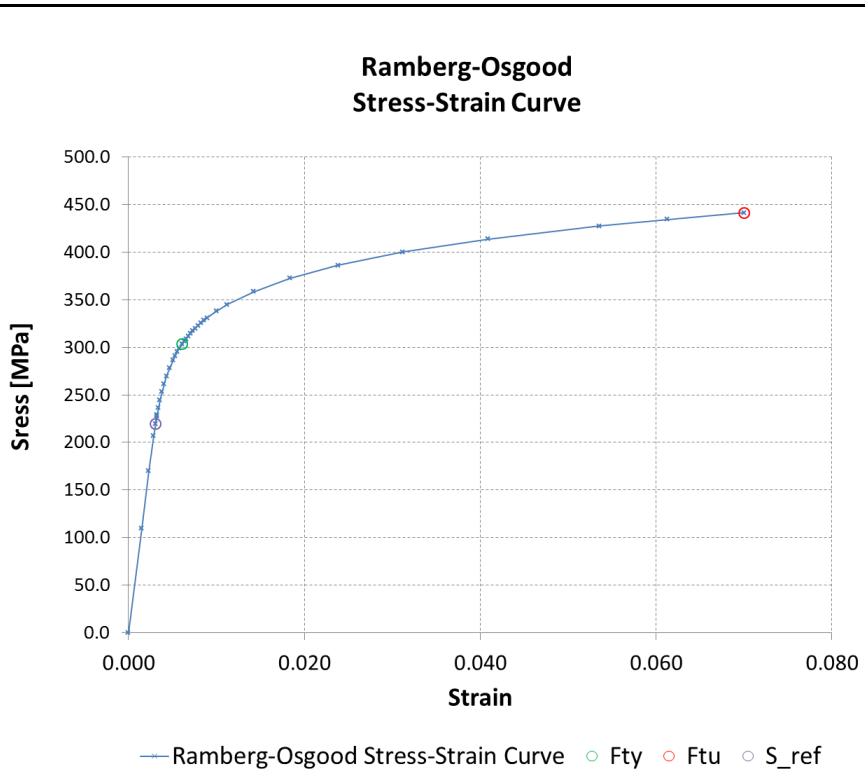


Table 3-1 Ramberg-Osgood Stress-Strain Curve

3.3.4.4. True Material Stress-Strain Curve

$$\varepsilon_{true} = ln(1 + \varepsilon^{nom})$$
$$\sigma_{true} = \sigma^{nom}(1 + \varepsilon^{nom})$$

σ^{nom} (MPa)	ε^{nom}
0.0	0.00000
109.9	0.00149
170.8	0.00232
207.6	0.00286
229.7	0.00325
220.1	0.00307
228.6	0.00322
237.1	0.00339
245.5	0.00358
254.0	0.00380
262.5	0.00404
271.0	0.00432
279.5	0.00466
288.1	0.00505
292.3	0.00528
296.6	0.00553
300.9	0.00580
305.3	0.00609
308.1	0.00630
310.9	0.00653
313.8	0.00677
316.6	0.00702
319.5	0.00729
322.4	0.00758
325.3	0.00788
328.1	0.00821
331.0	0.00855
333.9	0.00892
341.3	0.00994
348.6	0.01114
363.7	0.01414
379.2	0.01818
395.3	0.02357
412.4	0.03070
430.6	0.04004
450.4	0.05214
461.0	0.05943
472.2	0.06766

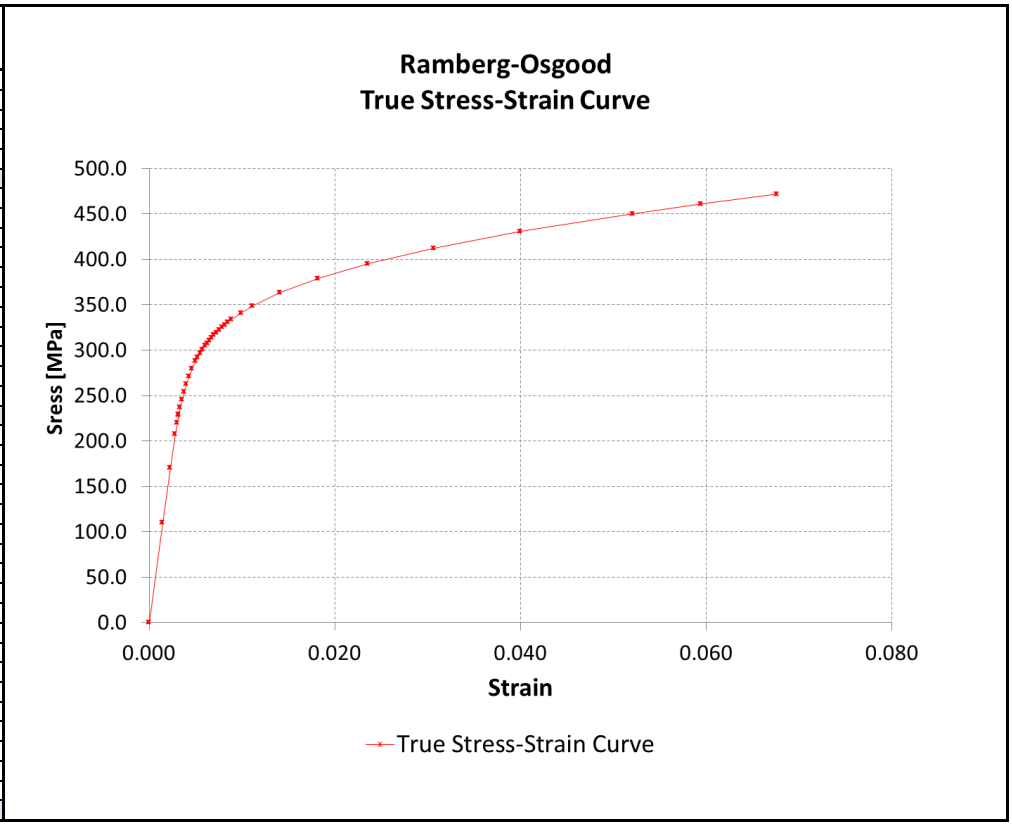


Table 3-2 Ramberg-Osgood True Stress-Strain Curve

3.3.4.5. True Material Stress- Plastic Strain Curve

$$\varepsilon_{true}^{pl} = \ln(1 + \varepsilon^{\text{nom}}) - \varepsilon_{true,\sigma_{ref}}$$

σ^{nom} (MPa)	ε^{nom}
220.1	0.00000
228.6	0.00015
237.1	0.00032
245.5	0.00051
254.0	0.00073
262.5	0.00097
271.0	0.00125
279.5	0.00159
288.1	0.00198
292.3	0.00221
296.6	0.00246
300.9	0.00273
305.3	0.00302
308.1	0.00323
310.9	0.00346
313.8	0.00370
316.6	0.00395
319.5	0.00422
322.4	0.00451
325.3	0.00481
328.1	0.00514
331.0	0.00548
333.9	0.00585
341.3	0.00687
348.6	0.00807
363.7	0.01107
379.2	0.01511
395.3	0.02050
412.4	0.02763
430.6	0.03696
450.4	0.04907
461.0	0.05636
472.2	0.06459

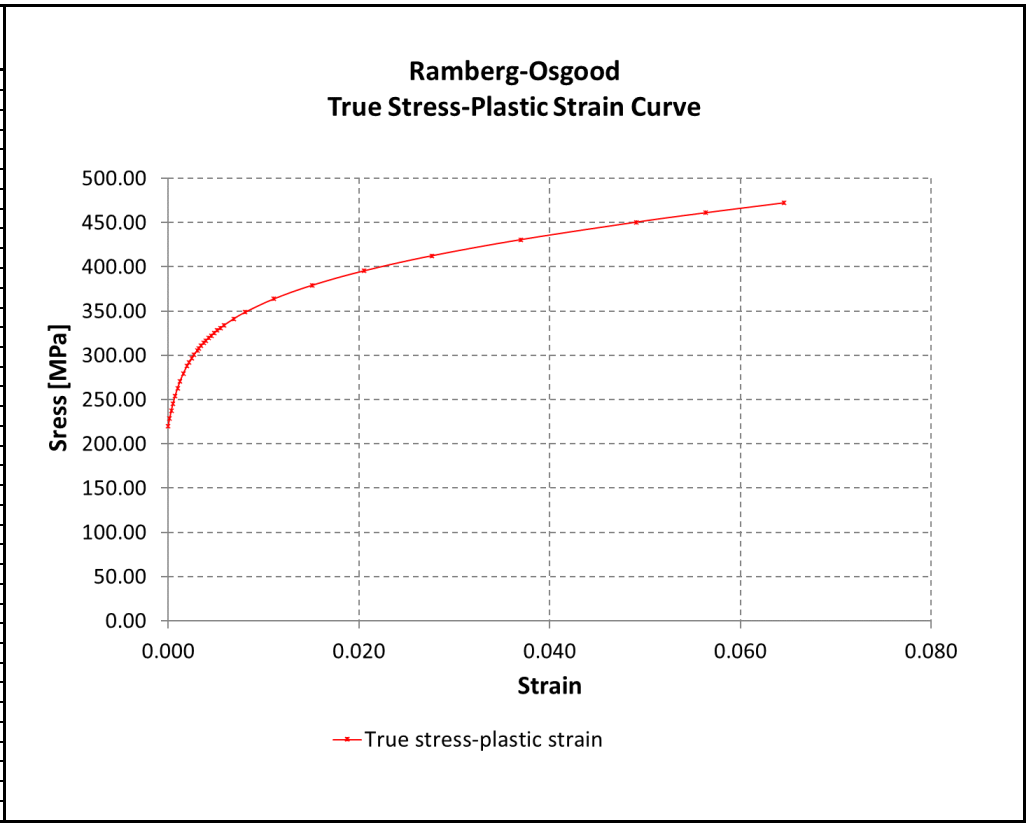


Table 3-3 Ramberg-Osgood True Stress-Plastic Strain Curve

3.3.4.6. ABAQUS Entries

```
**********************************************************************
** Material Input - Plastic Ramberg-Osgood Idealization
** Material Name: Al 2024 T351 - B - LT [25.43 < t < 38.10]
** E =            73774.0 MPa
** Ftu =          441.3 MPa
** Fty =          303.4 MPa
** e =            7 %
** n =            9.25
** Off. Strain =  0.01 %
** S_ref =        219.47 MPa
**********************************************************************
** Elastic Behavior
**********************************************************************
*Material, Name=Al 2024 T351 - B - LT [25.43 < t < 38.10]
*Elastic
73774.0, 0.31
**********************************************************************
** Elastic Behavior
**********************************************************************
*Plastic
220.14, 0.00000
228.60, 0.00015
237.06, 0.00032
245.53, 0.00051
254.00, 0.00073
262.49, 0.00097
271.00, 0.00125
279.52, 0.00159
288.07, 0.00198
292.35, 0.00221
296.64, 0.00246
300.94, 0.00273
305.26, 0.00302
308.09, 0.00324
310.94, 0.00346
313.79, 0.00370
316.65, 0.00395
319.51, 0.00422
322.38, 0.00451
325.26, 0.00481
328.15, 0.00514
331.04, 0.00548
333.95, 0.00585
341.25, 0.00687
348.63, 0.00807
363.67, 0.01107
379.18, 0.01511
395.35, 0.02050
412.40, 0.02763
430.62, 0.03697
450.39, 0.04907
461.00, 0.05636
472.19, 0.06459
**********************************************************************
```

3.3.5. Ramberg-Osgood Idealization – Sources of error

The Ramberg-Osgood idealization presents several sources of error, where the best-known error is derived from the input data that are available in material databases, where the plastic strain for Ftu does not match the failure strain, e.

Generally, the Ramberg-Osgood idealization presents good correlation in the first phase of the plastic curve up to Ftu. After the necking point, the R-O idealization does not match the right behavior.

Following figure shows a comparison between the real strain-stress curve of the material annealed Ti6Al4V sheet at room temperature (0.18 thick and A basis) and the Ramberg-Osgood idealization:

- E = 110316.2 N/mm^2 (Young Modulus)
- Ftu = 1006.6 N/mm^2 (Ultimate strength)
- Fty = 917.0 N/mm^2 (Yield strength)
- e_{max} = 8.0 % (Strain at rupture)
- μ = 0.31 (Poisson's ratio)

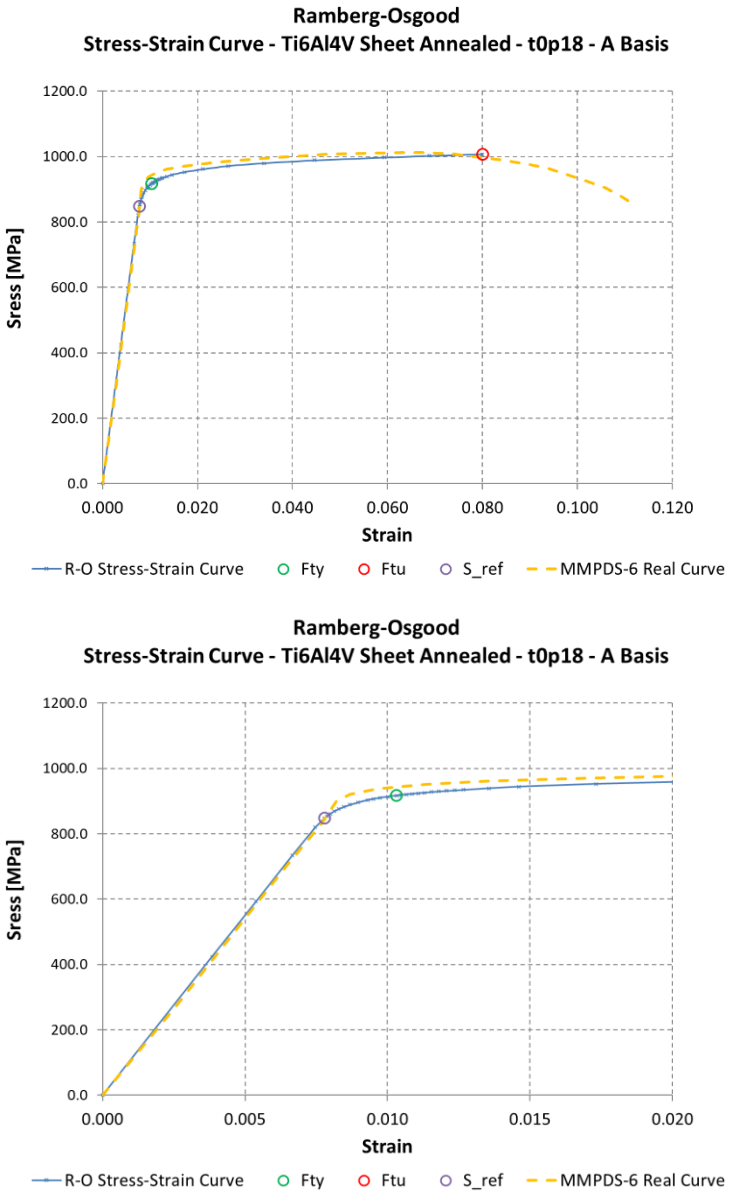


Figure 3-3 R-O vs Real Curve. Ti6Al4V Sheet Annealed - t0p18 - A Basis

4. DEFINING MATERIAL DATA IN ABAQUS ENVIROMENT

ABAQUS has no built-in system of units, and therefore, all input data must be specified in consistent units.

This is the reason why there are not material libraries loaded in ABAQUS environment.

If the system of units used for the analysis is always the same can be useful to modify the ABAQUS environment to define a material database with the common material used in analysis.

ABAQUS env file have to be modified with the following sentences:

```
def onCaeStartup():
   execfile('C:\\material.py')
```

Where, the “material.py” file contains the material database.

5. DFEM MODELLING GUIDELINES FOR NON-LINEAR ANALYSIS WITH ABAQUS (PLASTICITY)

5.1. SHELL AND SOLID ELEMENT TYPES

5.1.1. BENDING DOMINANT PROBLEMS

- SHELL
  - S4 if out-of-plane bending
  - S8 if in-plane bending (to avoid shear-locking effects)
- SOLID
  - C3D8I if very regular mesh (no distortion), C3D20
  - C3D10 (to be used without projection of edge nodes on the geometry)

5.1.2. CONTACT DOMINANT PROBLEMS WITHOUT BENDING

- SHELL
  - S4
- SOLID
  - C3D8
  - C3D10 (to be used without projection of edge nodes on the geometry)

5.1.3. CONTACT DOMINANT PROBLEMS WITH BENDING

- SOLID
  - C3D8I (if very regular mesh)

5.2. MESH DENSITY

5.2.1. STRESS DISCONTINUITY

Because the stress and strain are discontinuous between adjacent elements, the level of stress or strain discontinuity is a level of accuracy:

- Very accurate : d<5%
- Accurate : 5%<d<10%
- Medium : 10%<d<20%
- Coarse : 20%<d

5.2.2. FILLET RADIUS

To define peak stress or plastic behavior in fillet radius, the level of accuracy must be very accurate. It is necessary to define minimum 8 elements (90º) in the radius path.

Because the integration points in 3D solid elements are defined inside of element, stress and strain are calculated by extrapolating the data from the integration points to the nodes.

This extrapolation depends on element type (shape functions), and the accuracy results depend on mesh density and stress/strain gradient

It is recommended to use shell elements in the external side to achieve accurately the peak stress/strain and the plastic behavior.

6. POST-PROCESSING OF NON-LINEAR PLASTICITY ANALYSIS

6.1. RESERVE FACTOR

The post-processing of non-linear finite element plasticity analyses must be done with care, especially with respect to Reserve Factors calculation.

RF must be based on the following formula:

$$\text{R.F.} = \frac{\text{Allowable Load}}{\text{Applied Load}}$$

Where the Allowable Load is the load for which the criterion in allowable plastic strain is reached.

Allowable Load is obtained loading the FE Model with a given load level above Ultimate Load that produce the ultimate strength in the structure.

Following expressions for plastic analysis are also used, although they are not strictly right and non-conservative results could be obtained:

$$\text{R.F.} = \frac{F_{\text{tu}}}{\sigma_{\text{FEM}}}$$

$$\text{R.F.} = \frac{\text{elongation}}{\varepsilon_{\text{FEM}}}$$

6.2. PLASTIC ONSET FOR RAMBERG – OSGOOD IDEALIZATION

For Ramberg-Osgood material idealization, plastic onset is defined in a stress point below of F_{ty}. Therefore plastic strains are obtained in finite element analysis for stresses lower than F_{ty}.

Because these strains are lower than 0.002 (offset unit strain for F_{ty}), plastic behavior can be considered as negligible and elastic response can be assumed.

7. EXAMPLE

In this chapter a simply example has been developed in order to evaluate the influence of the different material idealizations and the ways to obtain the ultimate reserve factor.

7.1. GEOMETRY

Following figure shows the geometry analyzed:

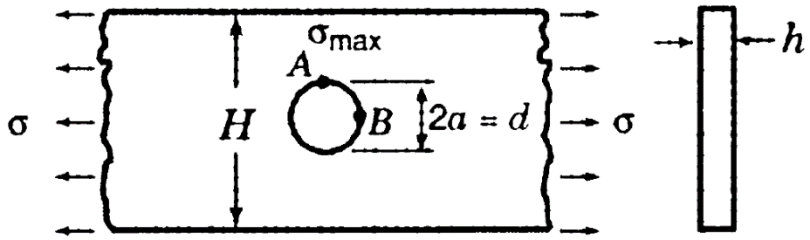


Figure 7-1 Geometry

Where:

- H = 60 mm
- D = 10 mm
- H = 5 mm

7.2. MESH

Linear hexahedra elements have been used, C3D8.

The hole surface has been meshed with shell element in order to achieve the peak stress accurately.

Following figure shows the mesh used for analysis:

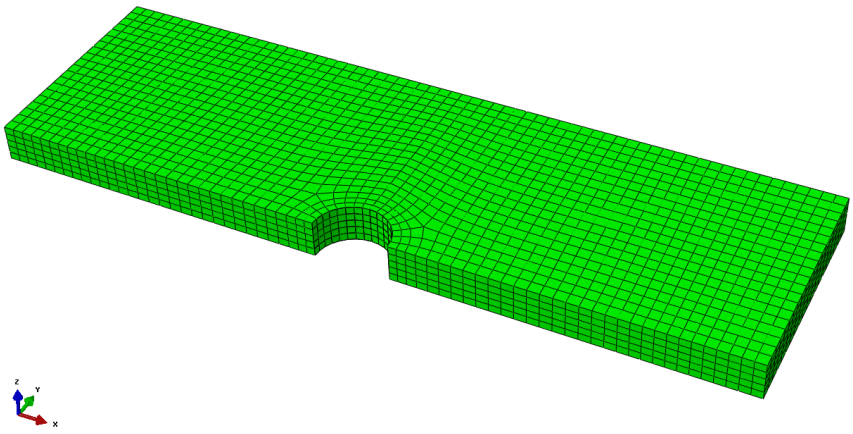


Figure 7-2 Mesh

Mesh quality has been validated for linear stress analysis with theoretical value:

- Theoretical results for a reference stress equal to 250 N/mm^2 (Peterson Chart 4.1) - σ_{peak,max_ppal} = 775,17 N/mm^2
- FEM Result - σ_{peak,max_ppal} = 779,2 N/mm^2
- Error = 0.53 %

7.3. MATERIAL

Material used for the example is Al 2024 T351 – B – LT – Range: [25.43 < t < 38.10].

The material properties are the following:

- E = 73774 N/mm^2 (Young Modulus)
- F_{tu}= 441.3 N/mm^2 (Ultimate strength)
- F_{ty}= 303.4 N/mm^2 (Yield strength)
- e = 7 % (strain at rupture)
- Off. Strain = 0.01 % (Plastic onset in Ramberg-Osgood curve)

7.4. LOAD

A reference stress of 250 N/mm^2 has been used for the example.

This stress has been applied as pressure in one side of the plate and it has been factorized to achieve maximum allowable load. Following factors has been used depending of material idealization:

- Elastic-Linear strain hardening – Factor = 1.2 → Applied stress in FEM = 300 N/mm^2
- Ramberg-Osgood – Factor = Factor = 1.3 → Applied stress in FEM = 325 N/mm2

7.5. BOUNDARY CONDITIONS

Following figure shows the boundary conditions used for the analysis:

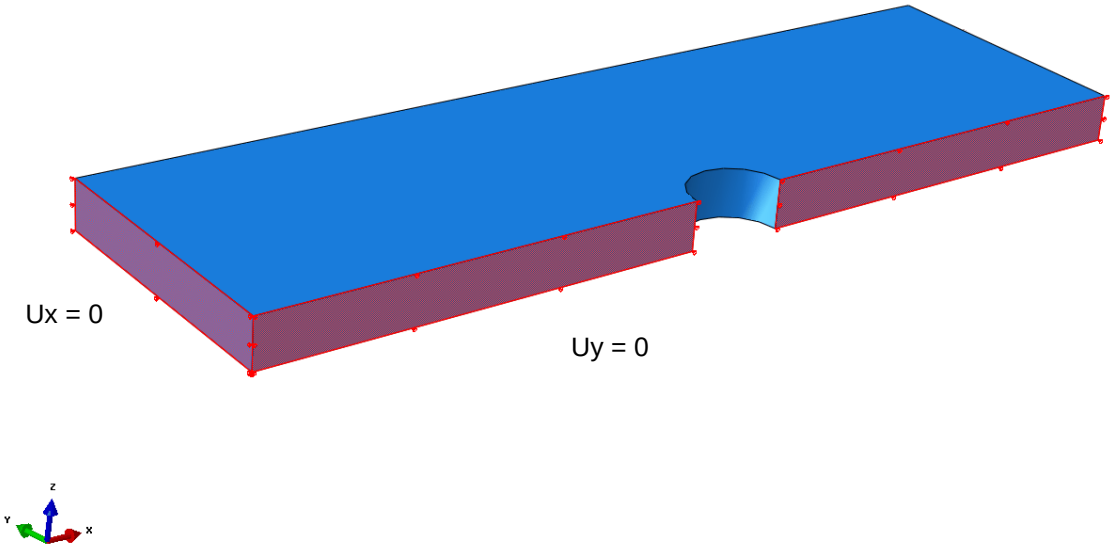


Figure 7-3 Boundary conditions

7.6. RESULTS

7.6.1. MODEL 01 – Material Idealization: Elastic-Linear strain hardening

$$\text{Applied_Load} = \text{Applied_Stress} \cdot \text{Area} = 250\text{N/mm}^2 \cdot 600\text{mm}^2 = 150000\text{N}$$
$$\text{Allowable_Load} = \text{FEM_Applied_Stress} \cdot \text{Area} \cdot \text{Failure_Load_Factor} =$$
$$\text{Allowable_Load} = 300\text{N/mm}^2 \cdot 600\text{mm}^2 \cdot 0.93 = 167400N$$
$$\text{R.F.} = \frac{167400\text{N}}{150000N} = 1.11$$

Where the Failure_load_factor is the ratio between the load for which the structural failure is achieved (σ_{fem} = F_{tu} for stress criterion) and the total load applied in FEM

Reserve factor obtained directly from stress is show as information. Note that this procedure is wrong and the reserve factor obtained could be not conservative:

$$\text{Peak_Stress} = 354.4\text{N/mm}^2$$
$$\text{Ultimate_Stress} = 472.2\text{N/mm}^2$$
$$\text{R.F.} = \frac{472.2\text{N/mm}^2}{354.4\text{N/mm}^2} = 1.33$$

7.6.2. MODEL 02 – Material Idealization: Ramberg – Osgood

$$\text{Applied_Load} = \text{Applied_Stress} \cdot \text{Area} = 250\text{N/mm}^2 \cdot 600\text{mm}^2 = 150000\text{N}$$
$$\text{Allowable_Load} = \text{FEM_Applied_Stress} \cdot \text{Area} \cdot \text{Failure_Load_Factor} =$$
$$\text{Allowable_Load} = 325\text{N/mm}^2 \cdot 600\text{mm}^2 \cdot 0.927 = 180765N$$
$$\text{R.F.} = \frac{180765\text{N}}{150000N} = 1.20$$

Where the Failure_load_factor is the ratio between the load for which the structural failure is achieved (σ_{fem} = F_{tu} for stress criterion) and the total load applied in FEM

Reserve factor obtained directly from stress is show as information. Note that this procedure is wrong and the reserve factor obtained could be not conservative:

$$\text{Peak_Stress} = 400.9\text{N/mm}^2$$
$$\text{Ultimate_Stress} = 472.2\text{N/mm}^2$$
$$\text{R.F.} = \frac{472.2\text{N/mm}^2}{400.9\text{N/mm}^2} = 1.17$$

7.6.3. Neuber Plasticity Correction

As additional information, Neuber plasticity correction is used to obtain the reserve factor in the example. Allowables used for Neuber correction are engineering values:

Linear Peak Stress (Von Mises) = 764.3 N/mm²

Corrected Stress Value = 398.0 N/mm2

$$\text{R.F.} = \frac{441.3\text{N/mm}^2}{394.5\text{N/mm}^2} = 1.11$$

7.6.4. Summary

MATERIAL IDEALIZATION	$\text{R.F.} = \frac{\text{Allowable Load}}{\text{Applied Load}}$	$\text{R.F.} = \frac{F_{tu}}{\sigma_{FEM}}$
Elastic-Linear strain hardening	1.11	1.33
Ramberg – Osgood	1.20	1.17

Neuber Plasticity Correction	---	1.11

Table 7-1 RF Summary