The Simulation of Different Types of Failure

POLITECNICO DI TORINO

AUTOMOTIVE ENGINEERING

Master's degree thesis

Supervisor: Prof. Luca Goglio

Candidate: Hu Jingchang

Academic Year: 2020/2021

Contents

The Simulation of Different Types of Failure ... 1

Introduction ... 3

Chapter 1: Tensile test ... 6

Theoretical model ... 6

Ductile material: ... 7

Brittle material: ... 11

Building model ... 12

For fracture simulation of ductile material ... 13

For fracture simulation of brittle material ... 23

Failure analysis ... 29

Ductile material ... 29

Brittle material ... 34

Chapter 2: Torsional test ... 41

Theoretical model ... 41

Ductile material ... 42

Brittle material ... 45

Building model ... 46

For fracture simulation of ductile material ... 46

For fracture simulation of brittle material ... 49

Failure analysis ... 53

Ductile material ... 53

Brittle material ... 58

Chapter 3: Conclusion ... 63

Chapter 4: Future work ... 65

Acknowledgement ... 66

Bibliography ... 67

Introduction

ANSYS has a wide range of applications in various fields but dealing with the failure problem is still a difficult point for ANSYS. This thesis will focus on the simulation of failure problem from basic tensile, bending and torsion tests and establish a method to deal with this problem.

In the traditional research and development process in the field of industrial product design and engineering, generally after the design is completed, prototypes are produced and the necessary strength, process rationality and functional verification are carried out. This traditional design mode has low efficiency and high cost. And every design change and product iteration will consume a lot of manpower and material resources. Therefore, in the field of modern industrial design and engineering, finite element modelling software is generally used for simulation in the early stage of product or engineering, such as process simulation, structural collision, multi-degree-of-freedom motion, etc., to qualitatively analyse whether the production process is reasonable and the strength, whether the design is sufficient, whether the product function meets the requirements, and others.

The principle of the finite element analysis method for structural problems is to use mathematical approximation methods to construct the geometry of the real physical system and simulate the load conditions. Through the transmission of load and deformation between simple and interacting cells, the complex infinite unknowns are simplified to finite interactions between cells. A suitable and simpler approximate solution is assumed for each cell, and various boundary conditions of the real physical model are applied to it, and then derive the total satisfaction conditions in the study domain to obtain the solution of the problem. Since for most practical problems it is difficult to obtain analytical solutions, at the same time, the finite element method has high calculation accuracy and can adapt to various complicated shapes, it can simulate the corresponding working environment by applying different boundary conditions and load conditions. In this stage, the engineering problems are simulated and preliminary analysed, so it has become an effective engineering analysis method.

ANSYS software is a relatively common finite element analysis software (Finite Element Analysis, FEA) in the industry at this stage. It can share and exchange data with most computer-aided design software, such as Creo, Nastran, I-DEAS, AutoCAD, etc. The large-scale general-purpose finite element analysis software integrating structure, fluid, electric field, magnetic field and sound field analysis is also one of the fastest growing CAE softwares in the world. ANSYS acquired the American company Fluent in 2006, which is a leader in the field of fluid simulation, and in 2008, acquired the American company Ansoft, which is a leader in the field of circuit and electromagnetic simulation. Through integration, ANSYS has become the world's largest simulation software company. The entire product line of ANSYS includes structural analysis (ANSYS Mechanical) series, fluid dynamics (ANSYS CFD (FLUENT/CFX)) series, electronic design (ANSYS ANSOFT) series, ANSYS Workbench and EKM, etc. Products are widely used in aviation, aerospace, electronics, vehicles, ships, transportation, communications, construction, electronics, medical, national defense, petroleum, chemical and many other industries.

ANSYS is powerful and easy to operate. Now it has become one of the most popular finite element analysis softwares in the world, ranking first in FEA competitions over the years. ANSYS software mainly includes three parts: pre-processing, analysis and calculation, and post-processing.

In the pre-processing module, users can perform solid modelling and mesh division, and users can also import models built in other engineering software. ANSYS mesh division mainly includes four methods: extended division, image division, free division and adaptive division. Users can choose the appropriate meshing method according to their needs and select the appropriate grid dispersion error for calculation. After the mesh is divided, users can apply loads to the research object, including boundary conditions and external or internal stress functions. Loads have different representations in different fields, but they can basically be divided into six categories: freedom constraints, force (concentrated load), surface load, body load, inertial load and coupled field load.

Once the model has been constructed, it is ready for calculation. ANSYS provides a basic calculation model, meanwhile, users can also build a calculation model through programming. ANSYS has limitations when studying the engineering tests of objects to simulate progressive failure problems. This paper will start from here to build simulation models for material failure problems in tension and torsion.

Once the simulation process has been completed, post-processing can be carried out. There are two types of data that can be processed by the post-processor: firstly, basic data, which refers to the degree of freedom solution obtained from each node. For the structural solution, is solved as a displacement field. For other types of solutions, there are the temperature for thermal solutions, the magnetic potential for magnetic solutions, etc. These result items are called nodal solutions. The second type is derived data, which refers to the result data derived from basic data. It is generally calculated for all nodes, all integration points, or derived data on the centroid of each cell, therefore, it is also referred to as cell solution. Post-processing of the data allows the accuracy of this model to be verified by the underlying physical phenomena, and the work behavior mode of this structure in real working conditions can be predicted through numerical simulation data.

When ANSYS is studying the failure phenomena of material under tension, compression and torsion, the existing models have some problems. In this paper, a computational model based on ANSYS will be constructed and its feasibility verified, starting from the basic tensile and torsional failure phenomena of brittle/ductile materials.

Chapter 1: Tensile test

Theoretical model

The tensile test is a basic material tension test in materials science and engineering, and the strength and plasticity indexes of metallic materials can usually be calculated from the directly obtained force-displacement curves. F-ΔL curves, which visualize the characteristics of material deformation and the relationship between the various stages of force and deformation. However, the quantitative relationship of the F-ΔL curve depends not only on the material but is also influenced by the geometry of the specimen. Therefore, tensile diagrams are often represented by nominal stress and strain curves (i.e., σ-ε lines):

Nominal stress of the specimen:

$$\sigma = \frac{F}{S_0}$$

Nominal strain of the specimen:

$$\varepsilon = \frac{\Delta L}{L_0}$$

S_0 and L_0 represent the area and the scale, respectively, under initial conditions. As shown in the figure below [1]:

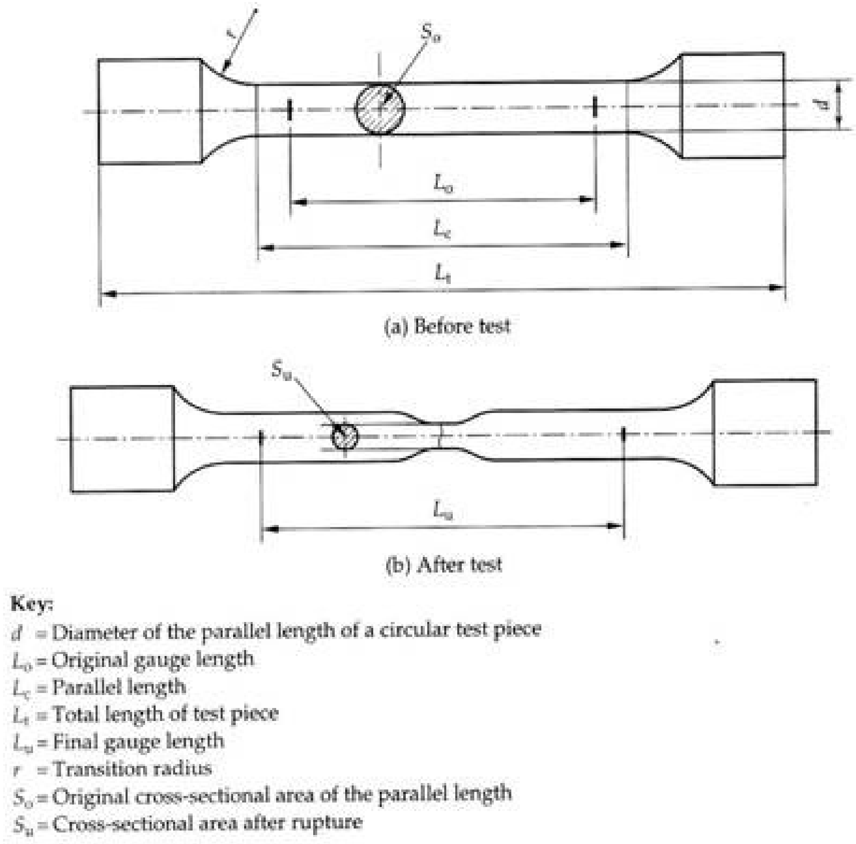


FIG 1

σ-ε curves are similar to F-ΔL curves but eliminate the effect of geometric dimensions and are therefore representative of the material properties. It is on the σ-ε curve that the mechanical properties of some materials under unidirectional tensile conditions are defined. If the test provides an accurate tensile diagram, then the main mechanical property indices under unidirectional tensile conditions can be accurately determined.

Ductile material:

Take mild steel as an example, when doing tensile test, the state of ductile material can be roughly divided into four stages: elastic stage, yield stage, strengthening stage, and local deformation stage.

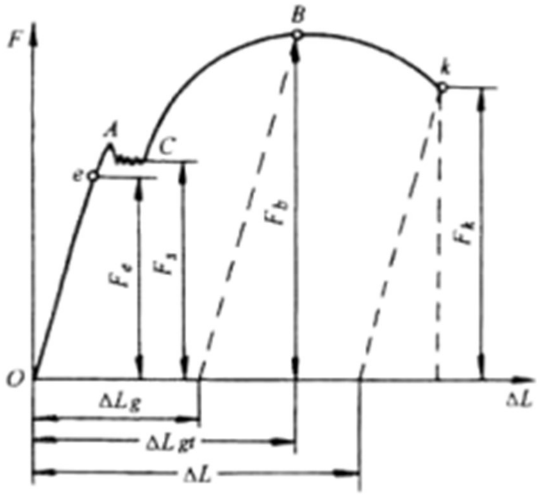


FIG 2

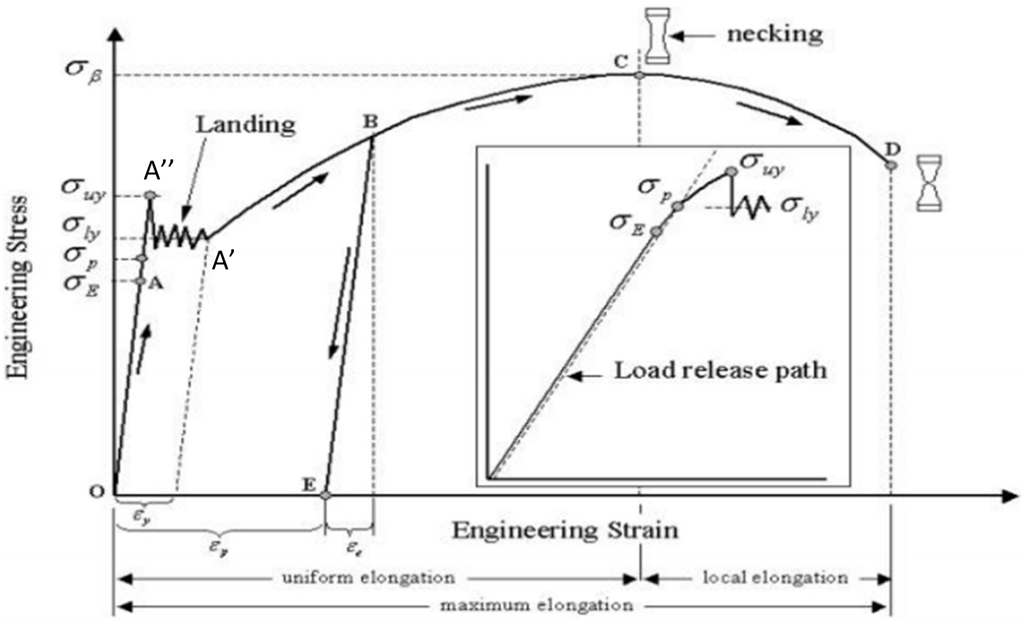


FIG 3 [2]

As FIG 3 shown above, in the OA section, the magnitude of the force applied to the specimen and its elongation are linearly related, at which point, if the force applied to it is removed, the specimen will return to its original size without any residual deformation. This stage is the elastic stage of the ductile material, and it is customary to consider that the material obeys Hooke's law in the elastic range, and its stress-strain is proportional to the relationship.

$$\sigma = E\varepsilon$$

The scale factor E represents the slope of the straight line OA and is called the modulus of elasticity of the material.

In the AA' section, as the elongation grows, the force on the specimen remains constant within a certain fluctuation range, which indicates that the material temporarily loses its ability to resist further deformation. The point A'' is the upper yield limit of the material, which marks the beginning of plasticity. This stage is known as the yielding stage. Once the stress in a structure or part exceeds the σ_{ly}, the material will yield and the part will fail due to excessive deformation. Therefore, the lower yield limit σ_{ly} is often used as the basis for determining the allowable stress when designing for strength.

In the A'C section, the specimen continues to elongate and the force applied continues to rise slowly, the material regains its resistance to continued deformation, and the load grows. If unloaded at this stage, the elastic deformation will then disappear, while the plastic deformation will remain forever. The unloading path in the strengthening phase is parallel to the elastic phase. If reloaded after unloading, the loading line is still parallel to the elastic phase, but after reloading, the elastic phase of the material lengthens and the yield strength increases significantly, while the plasticity decreases accordingly. This phenomenon is called as deformation strengthening or cold work hardening. Cold work hardening is one of the most valuable properties of metallic materials. The combination of plastic deformation and deformation strengthening is an important means of strengthening metallic materials. For example, shot blasting, extrusion, cold dialing and other processes, is the use of cold work hardening of materials to improve the strength of the material. The plastic deformation in the strengthening stage is uniformly distributed along the axial direction. C point is the highest point of the σ-ε curve, defined as the strength limit of the material and also known as the tensile strength of the material, recorded as σ_β. For ductile materials (such as mild steel) σ_β is the maximum resistance to uniform plastic deformation of the material, is the mark of the material into the necking stage.

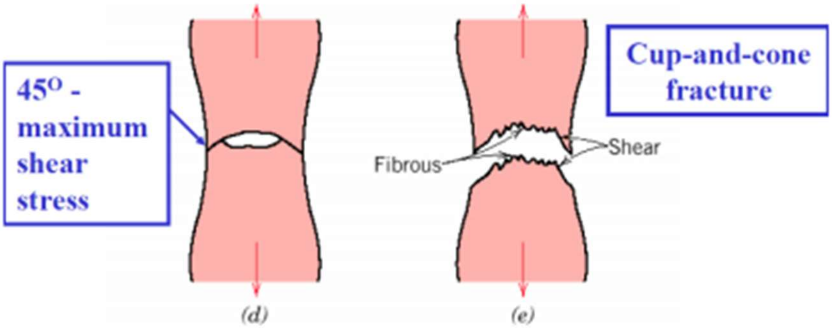
The CD section is the necking stage, after the stress reaches the strength limit, plastic deformation begins to take place locally. The local section shrinks sharply, the bearing area decreases rapidly, and the load on the specimen decreases quickly until fracture. At fracture, the elastic deformation of the specimen disappears, and the plastic deformation is left on the broken specimen.

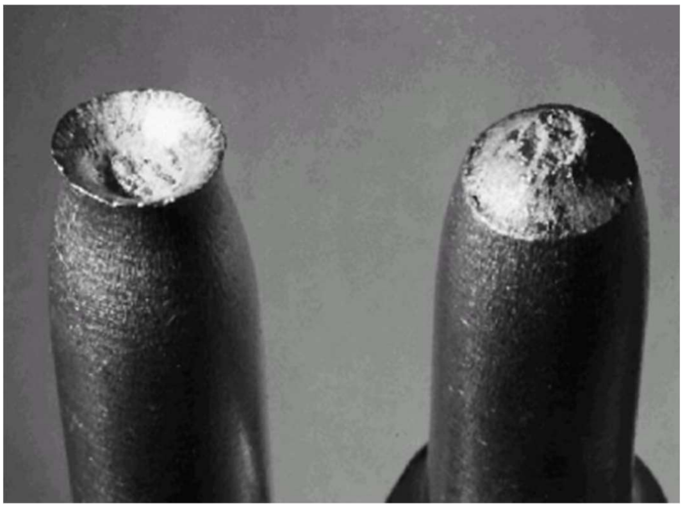
It is generally accepted that materials are defective and these defects are contributors to the stress concentration. When the material is under tension or pressure, a large number of dislocations will plug near the defects (e.g., grain boundaries, second-phase particles, etc.), and when the density of plugged dislocations reaches a maximum, new dislocation plugging is no longer generated and the stress field near the defects reaches a maximum, and the work hardening is less than the increase in external stress. Then in the next deformation, the release of stress near the defect is achieved by the emission of dislocations. The necking phenomenon is thus generated.

From the above analysis, it is clear that the maximum stress on the specimen will be concentrated at the necking, so fracture usually occurs first by the necking, which is one of the most important phenomena that can be observed in tensile testing of ductile materials. However, in computer simulations, where the material cell is set to perfect by default without adding material defects, the necking phenomenon, and consequently fracture, could theoretically occur at any location in the specimen. Therefore, in order to control the fracture in a place where it can be easily observed, we need to make minor adjustments in the shape of the specimen, which will be specifically mentioned in the next part of the model presentation.

Since the maximum stress is always concentrated at the necking area, the fracture starts from there as well.

Specifically, in the necking region, the stress decreases from the center to the periphery, so that the fracture condition is always reached first in the center. After the center fracture, plastic deformation continues to occur at the periphery until the periphery also reaches the fracture condition, at which time the whole specimen is fractured. Because of this fracture sequence in ductile materials, the fracture is cup-shape. As shown in the figure below, mild steel, a typical ductile material, has a 45° shear lip at the periphery of the fracture, and the fracture tissue is dark gray fibrous.

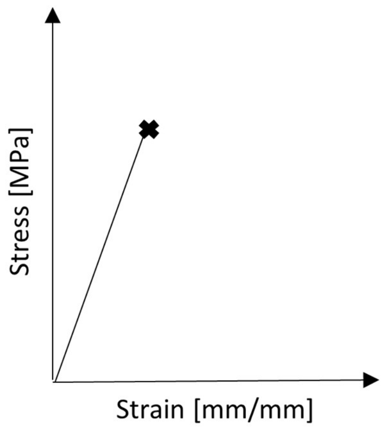




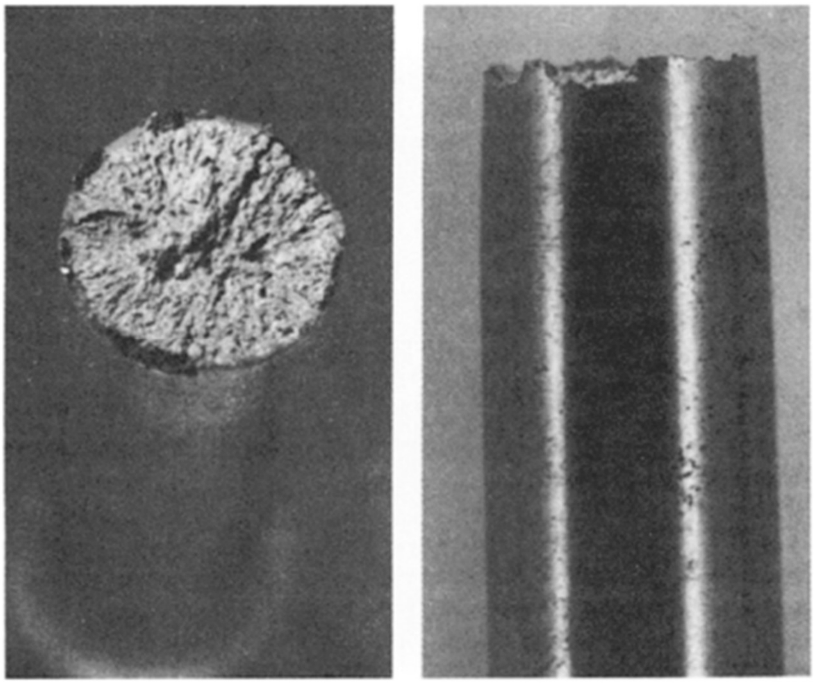
CUP-AND-CONE FRACTURE IN ALUMINUM

Brittle material:

The tensile process of brittle materials is simpler than that of ductile materials, and can be approximated as a direct transition to fracture via the elastic phase.



Its damage fracture along the cross-sectional direction, indicating that the fracture of cast iron is caused by tensile stress, its strength index is only σ_u. As seen by the tensile curve (as shown above), brittle material elongation after fracture is very small, so often in the absence of any forewarning of the sudden occurrence of brittle fracture. Therefore, if these materials are used improperly, accidents are very likely to occur. As shown in the figure below, cast iron as a typical brittle material, the fracture is perpendicular to the direction of the positive stress, and the cross-section is flush with the shiny crystalline organization, which is typical of brittle fracture.



BRITTLE TENSILE FRACTURES [3]

Building model

Before simulation, we must clarify that ANSYS Workbench has some limitations in dealing with fracture problems, i.e., ANSYS by default will not break the specimen no matter how much force is applied to it.

The reason for this problem lies in the algorithm. The default ANSYS algorithm is the implicit algorithm for solving the KX=F equation, which is more accurate, but it cannot calculate effects such as fracture, while the corresponding explicit algorithm can determine whether the material is pulled or not based on the results of each step. Explicit dynamic and ls dyna are available in ANSYS Workbench for calculating the explicit algorithm, and both of them are available for calculating tensile fracture. However, the main problem with both is that the default calculation requires a complete material-related theory and fracture criterion, which is not easy to use for some materials that do not have a relevant theory. Therefore, in this paper, the APDL command is used to simulate fracture, i.e., to insert a segment of the parametric design command to simulate fracture without a specific practical fracture criterion.

For fracture simulation of ductile material

1. **Material.**

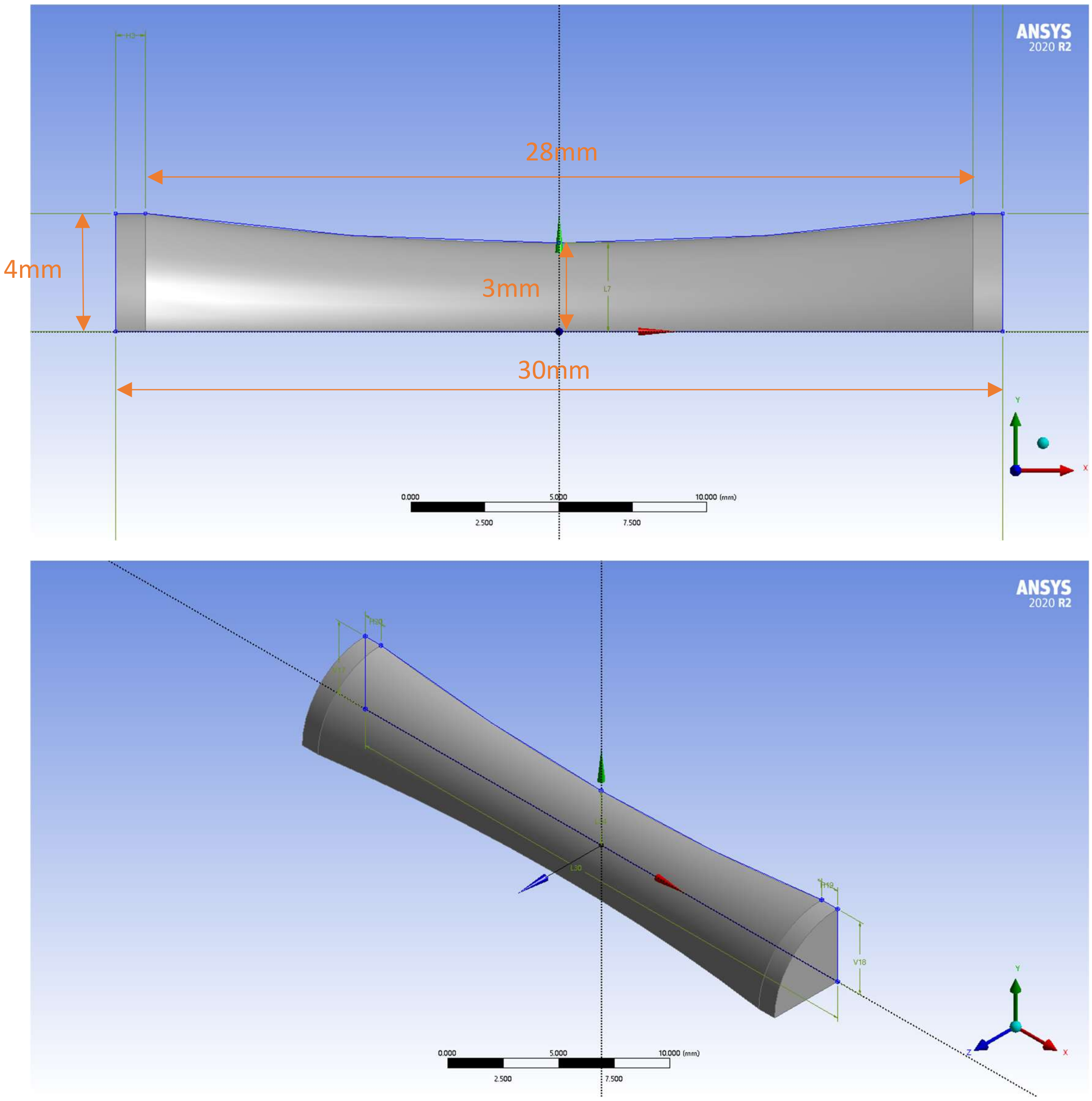
   In this paper, the Structural Steel NL provided by ANSYS has been used where ductile materials are used, and data relating to the corresponding isotropic elasticity data has been added to it. The specific parameters for this material are shown in the table below.

Property		Value	Unit
Density		7850	$kg \cdot m^3$
Isotropic Elasticity	Young's Modulus	2E+11	Pa
	Poisson's Ration	0.3	/
Bilinear Kinematic Hardening	Yield Strength	2.5E+08	Pa
	Tangent Modulus	1.45E+09	Pa

TABLE 1

2. **Geometry.**

   As real specimens are not perfect models, fracture usually occurs first at defective cells, but metal specimens in computer models always have a perfect crystal structure, so if the specimen is not treated, fracture will occur at the ends where the boundary conditions are applied. Besides, it is not easy to observe the shape of the fracture surface. The approach taken in this paper is to adjust the radius of the cross-section of the column so that the radius of the cross-section in the middle part is minimized. As the cross-sectional area is minimized, the stress on the middle part is greatest when subjected to the same magnitude of force, which also results in a necking phenomenon similar to that caused by cell defects, and thus the fracture is controlled to where we want it to be. At the same time, in order to observe the necking phenomenon, the surface of the column prior to stretching must not be significantly depressed, so only very minor changes to the cross-sectional diameter of the column model can be made in this work. In addition, as the shape of the column is uniform and the forces are uniform, the whole force and fracture process is axisymmetrically distributed, and 1/4 column can be used to reduce the amount of operations. Based on the above considerations, the finalized geometric model and correlation is shown below.

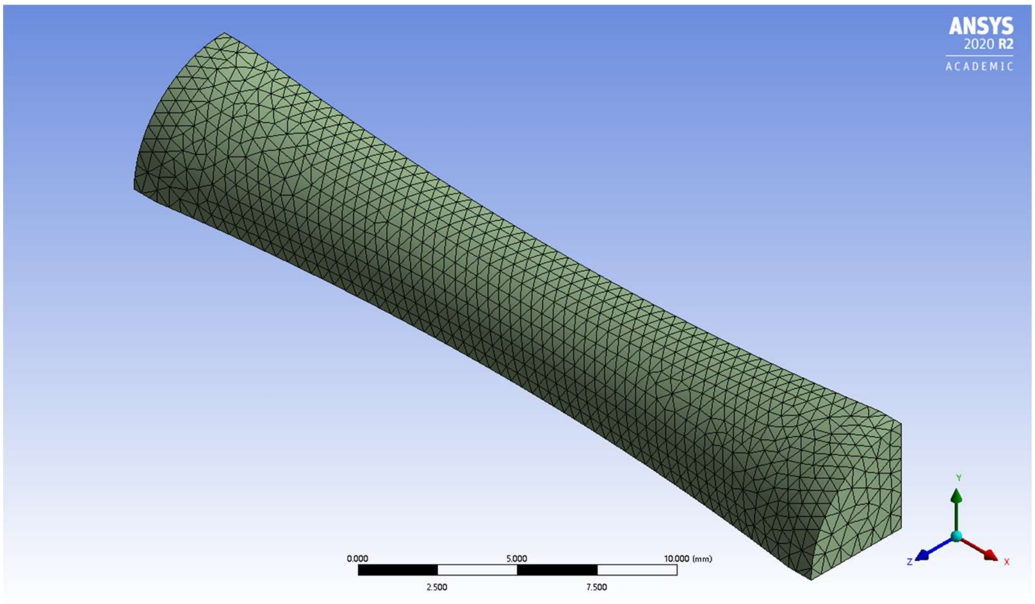


GEOMETRY

3. **Meshing.**

This paper uses the free student version of ANSYS, for which ANSYS gives a maximum calculation limit of 32,000 points. Due to this limit, none of the model meshes in this paper can be made very fine, but some improvements can be made based on this limit. By using the sizing method, sphere of influence function in ANSYS meshing and setting the appropriate radius of influence and growth rate, the finest mesh size can be distributed in the center of the column which is the part of interest to us.

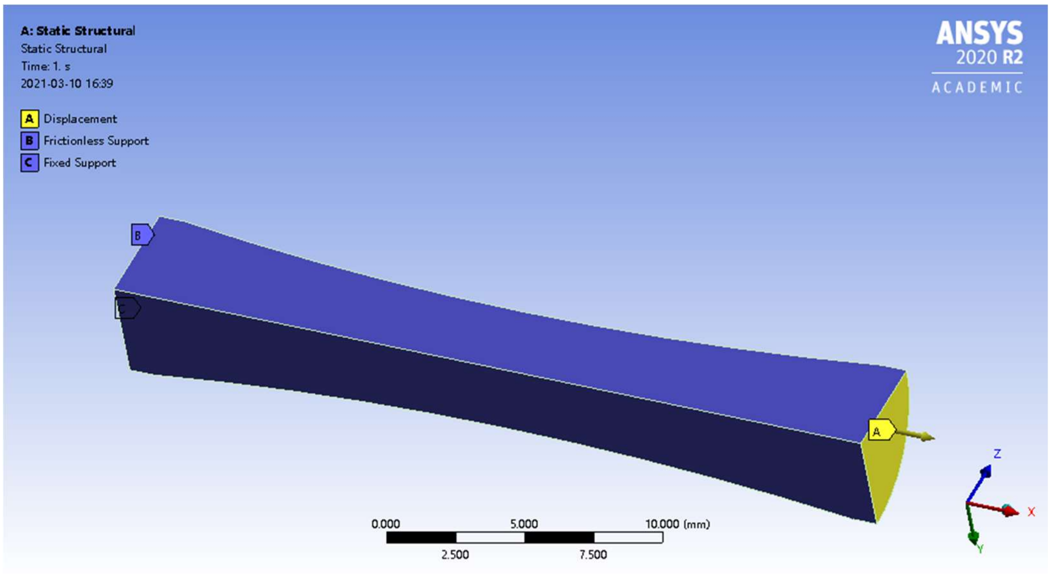
However, because the inhomogeneous mesh has a great impact on the convergence of the calculation, in this case after a long time and a lot of debugging, the results still cannot converge. Therefore, in the tensile test simulation of ductile materials, the mesh division using the overall uniform mesh to achieve convergence. The meshing in this case is shown in the figure below.



MESHING

4. **Boundary conditions.**

In order to simulate the tensile test where one end of the specimen is held in a fixture and the other end is stretched at a constant speed in the fixture, the boundary conditions are applied to the end faces of the column in this case. As shown in the figure below, a fixed support is set for the left end of the specimen and a displacement is set for the right end. As the use of the symmetry tool in ANSYS software increases the amount of calculations, it is possible to simulate axisymmetry in the tensile test by using frictionless support. As shown in the picture below, by applying frictionless supports to both sides, the two sides are unable to produce radial displacements, thus simulating a 1/4 column with only axial tension.



BOUNDARY CONDITIONS

5. **APDL command.**

APDL stands for ANSYS Parametric Design Language. It is an interpreted language like FORTRAN and provides general programming language features such as parameters, macros, scalar, vector and matrix operations, branching, looping, repetition and stabilization of the ANSYS finite element database. In addition, it provides simple interface customization for interactive parameter input, message mechanisms, interface drivers and running applications. By using the APDL programming language and macro technology to organize and manage the ANSYS FEA commands, parametric modelling, application of parametric loads and solutions and display of parametric post-processing results can be achieved, thus enabling the whole process of parametric FEA. In addition, APDL is the basis for design optimization in ANSYS by creating a parametric analysis flow in which design parameters can be optimized and improved to achieve the goal of optimal design. [4]

Specifically, in this case, without the use of APDL, if a displacement greater than the actual value is given to one end of the specimen, ANSYS Workbench's default algorithm will cause it to be subjected to stresses beyond the ultimate yield point while continuing to stretch. In order to make the simulation as close to the real situation as possible, we need to insert an APDL command into the default ANSYS algorithm so that the specimen will fracture when subjected to a stress greater than the ultimate yield point.

The main part of the APDL command in this example uses a command posted on the ANSYS forum by peteroznewman [5], based on which the material failure criterion is changed. The logic of this APDL command is that the whole operation is divided into a number of steps. In each step, the equivalent stress is calculated for each element in the specimen and compared to the failure criterion (in this case, the failure criterion is the ultimate yield stress for structural steel NL). If it is greater than the failure criterion, the element is killed using the ekill command, otherwise the operation continues to the next step.

In ANSYS, the ekill command is used to deactivate an element. A deactivated element remains in the model but contributes a near-zero stiffness (or Any solution-dependent state variables (such as stress, plastic strain, creep Any solution-dependent state variables (such as stress, plastic strain, creep strain, etc.) are set to zero [6]. In this case, the element that is subjected to stress greater than the ultimate yield stress is killed and cannot be displayed in the results, so that the fracture state can be simulated.

As shown in the script below, arg1 represents for the stress when the specimen is failed and arg2 is the number of steps we want ANSYS to calculate. Both of them are defined in the “details” box on the left. The command “s,eqv” represents for equivalent stress.

The fracture state of a ductile material can be successfully simulated using such a loop command. However, since the command kills all elements within a range, rather than the element that is subject to the maximum stress within that range, when the specimen is stretched beyond its fracture-bearing length, too many elements will be killed due to stresses beyond the maximum point, resulting in unreasonable fracture results. The biggest difficulty in using this approach is therefore how to obtain the most accurate fracture displacement through extensive debugging and in turn apply it as a boundary condition at one end of the specimen.

```
fail_stress=arg1
steps=arg2

*get,TIME_END,common,,stepcm,,real,2
timeinc=TIME_END/steps
time,timeinc
solve

*do,ICOUNT,1,steps-1

  /post1
  set,last

  cmsel,s,killelem
  etable,MYPLAS,s,eqv
  esel,r,etab,MYPLAS,fail_stress,500
  *get,ECOUNT,elem,,count

  cm,MYELEM%ICOUNT%,elem
  finish
allsel
cmwrite,temp%ICOUNT%,cm
parsav

  /solu
  antype,,rest
  parres,change

*do,j,1,ICOUNT
/gopr
   /input,temp%j%,cm
   *GET, exists, COMP, MYELEM%j%, TYPE
   *if,exists,ne,0,then
     cmsel,s,MYELEM%j%
     elist
     ekill,all
   *endif
*enddo

  allsel,all
  time,timeinc*ICOUNT+timeinc
  solve
  finish
*enddo
```

6. Debugging.

Some settings in the software operations can be changed in ANSYS by adjusting the **Analysis Settings**. For nonlinear analysis, the "large deflection" setting needs to be turned on. Since the restart statement is programmed into the APDL command, you need to change the generate restart points to manual mode here. You can also select the expected output values in the output controls as needed.

⊟	**Solver Controls**	
	Solver Type	Direct
	Weak Springs	Off
	Solver Pivot Checking	Program Controlled
	Large Deflection	On
	Inertia Relief	Off
	Quasi-Static Solution	Off
⊞	**Rotordynamics Controls**	
⊟	**Restart Controls**	
	Generate Restart Poi...	Manual
	Load Step	Last
	Substep	Last
	Retain Files After Ful...	No
	Combine Restart Files	Program Controlled
⊞	**Nonlinear Controls**	

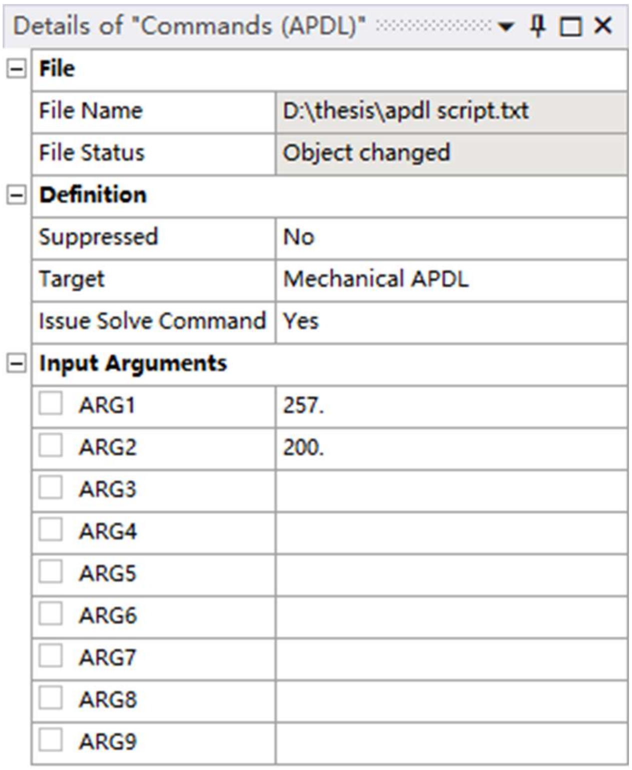
ANALYSIS SETTINGS

Since the material used in this paper is a bilinear material, there is only one point of inflection and therefore no yielding phenomenon can be observed. The only inflection point is designated as the yield point, and due to the lack of a fracture criterion, a general assumption is made about the location of the fracture point in this paper. Since the purpose of this paper is to give a general solution for fracture simulation in the absence of a fracture criterion, this fracture point only needs to be chosen to be larger than the yield point. In this example, the rightmost point of the σ-ε curve given in the material card is used as the failure point, i.e. equivalent stress=257MPa and total strain=0.00625.

BILINEAR HARDENING DATA OF STRUCTURAL STEEL NL

As mentioned earlier, according to the circular logic of the APDL command used in this thesis, a failure criterion needs to be selected as the judgment condition for the elements to be killed, i.e., the engineering data corresponding to the above fracture point. Theoretically, we can achieve the same purpose by using Von Mises equivalent stress or equivalent plastic strain or equivalent total strain as the failure criterion. In this paper, Von Mises equivalent stress is chosen.

The next step is to set the failure criterion in the APDL command to Von Mises equivalent stress, and set a fixed value for it, in this case 257 MPa. As explained in the previous section on APDL command, here the value 257 is assigned to arg1. The value assigned to arg2 represents the step of the calculation, and it is easier to get converged results by setting it as large as possible within the allowed range of operations. Here in this case, 200 is assigned.



Details of "Commands (APDL)"

File	
File Name	D:\thesis\apdl script.txt
File Status	Object changed
Definition	
Suppressed	No
Target	Mechanical APDL
Issue Solve Command	Yes
Input Arguments	
ARG1	257.
ARG2	200.
ARG3	
ARG4	
ARG5	
ARG6	
ARG7	
ARG8	
ARG9	

APDL COMMAND ARG SETTINGS

However, before running the APDL command, there is one more preparation that must be done to find the elongation of the specimen when it is just about to fracture. This is done by:

a. In Analysis Setting, set “define by” to **substeps** and set a slightly large number for it. There are three items in the substeps setting: the **initial substeps** must be greater than or equal to the **minimum substeps**, and the **maximum substeps** can be set as high as possible to obtain converged results. This operation can break down the process into multiple steps to obtain more accurate results. In this paper, these three items are set to 50, 50 and 1000, meaning that the 1-second stretching process is divided into at least 50 steps. Therefore, the results of every 1/50 seconds for the entire stretching process can be obtained.

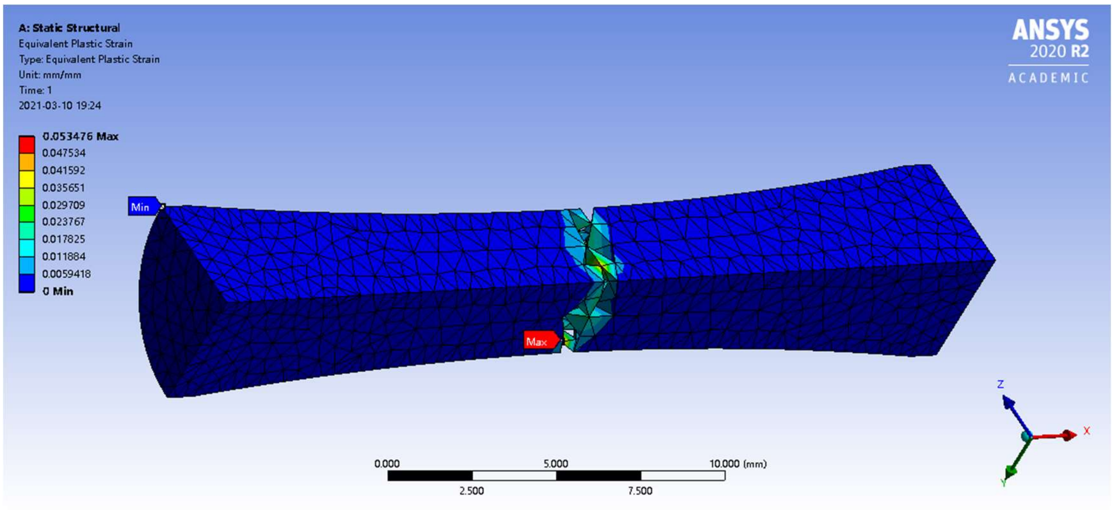
Details of "Analysis Settings"	
Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	On
Define By	Substeps
Initial Substeps	50.
Minimum Substeps	50.
Maximum Substeps	1000.

STEPS CONTROL

b. Set a large displacement for the boundary condition. This step makes use of the implicit algorithm of ANSYS, i.e., the specimen continues to be stretched and does not fracture when its maximum bearing range is exceeded. Therefore, we can be sure that the fracture must occur before the final displacement is achieved. In this paper, 0.5 mm was set as a foreseen value for the tensile displacement.

c. In the result, we find the time point of "fracture" and get the amount of elongation at the corresponding time point. As shown in the figure below, the result of equivalent stress is divided into xx parts, and the closest time point in this process is found to be greater than 257MPa. Against this time point, the corresponding displacement can be found in the deformation result at the stretching end. The displacement of the specimen at the time of fracture is approximately around this number.

After obtaining the stretching amount at the fracture point, this value is input as the displacement amount at the stretching end, and then the APDL command is activated to restart the operation. If the result does not converge, fine-tune the displacement downward; if the specimen is not pulled off in the result, fine-tune the displacement upward. In this way, the fracture simulation results can be obtained as expected.



DUCTILE FAILURE OF TENSILE TEST

For fracture simulation of brittle material

1. Material.

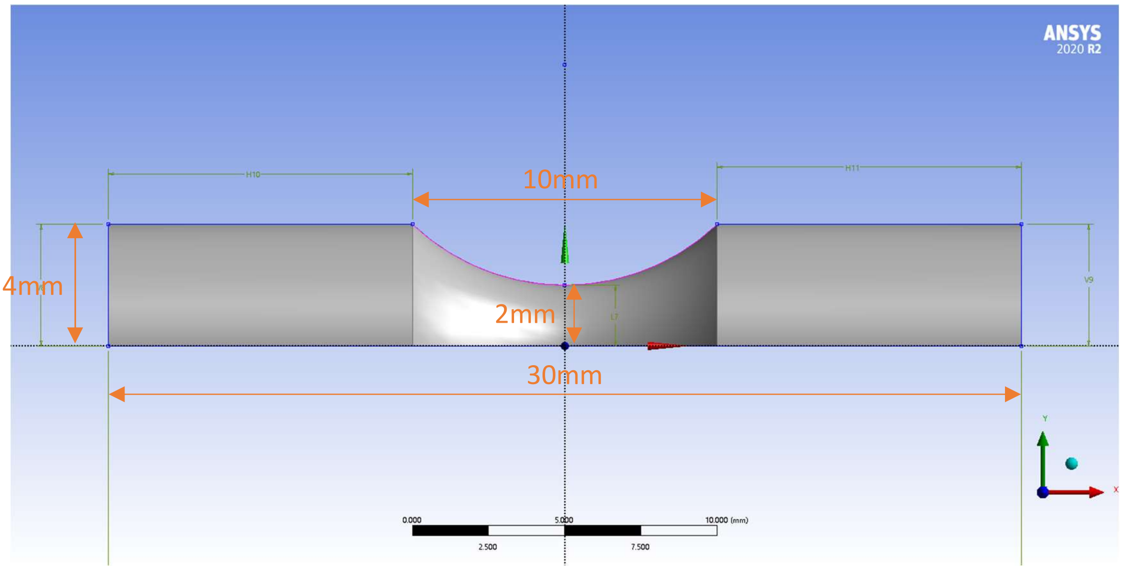
In this paper, all cases using brittle materials use the gray cast iron provided by ANSYS, which has a maximum tensile strength of 240 MPa. The specific data are as follows.

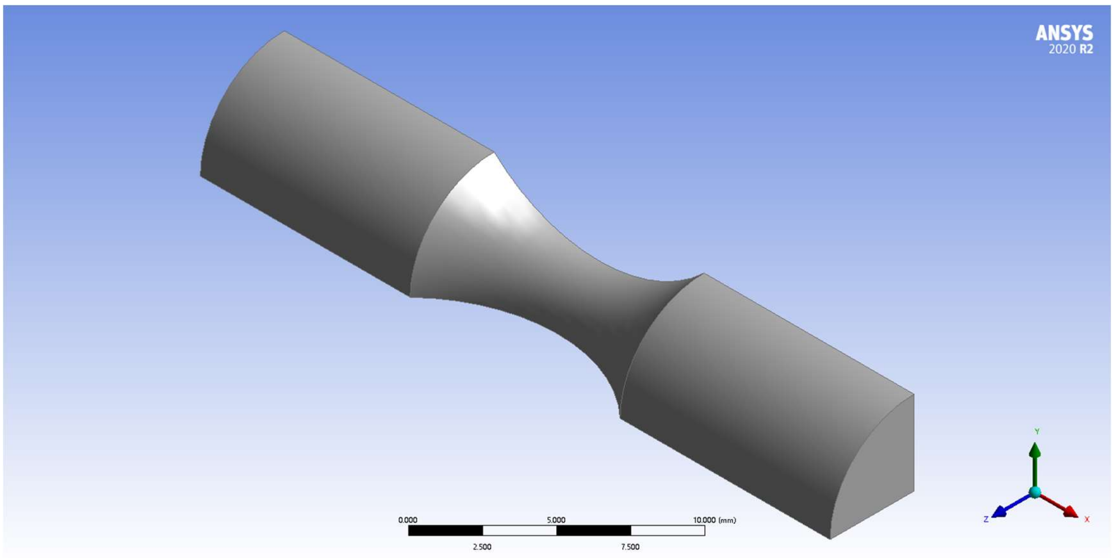
property		value	unit
Density		7200	$kg \cdot m^3$
Isotropic Elasticity	Young's Modulus	1.1E+11	Pa
	Poisson's Ratio	0.28	/
Tensile Yield Strength		0	Pa
Compressive Yield Strength		0	Pa
Tensile Ultimate Strength		2.4E+08	Pa
Compressive Ultimate Strength		8.2E+08	Pa

TABLE 2

2. Geometry.

Based on the above theoretical analysis, it is known that there is no necking in the fracture process of brittle materials, so there is no need to control the radius of the column cross section to a small difference in the fracture of brittle materials. The same problem is faced with the simulation of ductile materials. Since the metal cell in the software has a perfect structure by default, a hourglass shaped specimen will help in the simulation of brittle materials in order to control the fracture location where we want it, as shown in the figure below. This geometry not only controls the fracture at the smallest cross-section, but also makes it easier to adjust the mesh without causing the results to fail to converge due to the inhomogeneity of its outer surface. Similarly, to observe the stress-strain variation of the specimen from the periphery to the center, a 1/4 model is also used for the torsional model in brittle case.





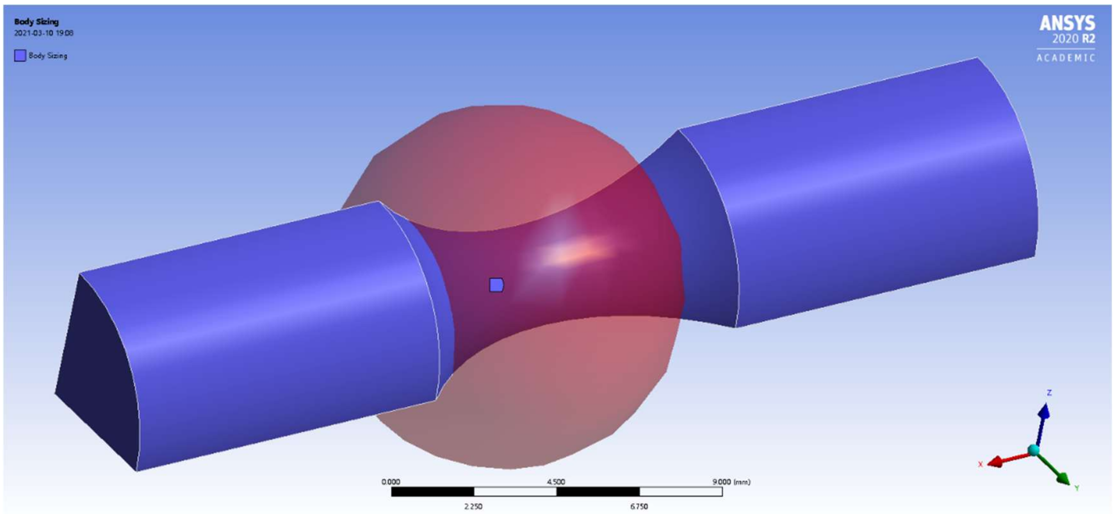
GEOMETRY

3. Meshing.

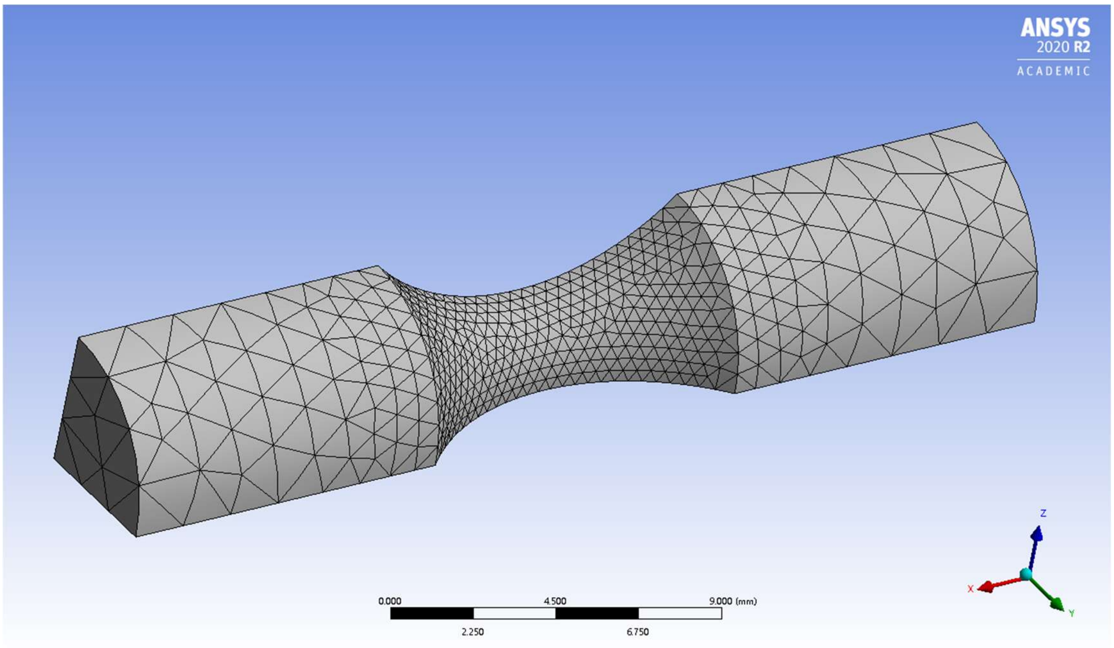
In order to get the most accurate results possible within the limits of ANSYS Student, it would be a good idea to set the finer mesh at the place of interest. In ANSYS you can use the sphere of influence type of body sizing to control the finer mesh within the spherical area shown below. By adjusting the growth rate, the mesh can be adjusted to be more uniform and easier to calculate. In this case, growth rate is set to 1.2.

Details of "Body Sizing" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Sphere of Influence
Sphere Center	Global Coordinate System
Sphere Radius	5.0 mm
Element Size	0.4 mm

BODY SIZING DATA



SPHERE OF INFLUENCE



MESHING

4. **Boundary conditions.**

Since the fracture simulations are performed under the same conditions, the boundary conditions are also fixed at one end and a certain amount of displacement is applied to the other end. As with the fracture simulation of ductile materials, a more accurate amount of displacement still needs to be found by calculation.

As mentioned earlier, the search for the amount of stretching at the brittle fracture point requires the determination of a fracture condition. Due to the complexity of the physical nature of metallic materials, their brittle fracture has been a research difficulty that has attracted much attention. A lot of research has been conducted on the brittle fracture of cracked and uncracked bodies, and a large number of guidelines and models have been used to explain different brittle fracture test phenomena, among which the most famous ones are the maximum circumferential stress theory [7] and the strain energy density factor theory [8] proposed by Sih G C for cracked bodies.

Since the fracture criterion is not the subject of this thesis, the purpose of this thesis is to propose a generalized method to simulate fracture, we do not dwell on the relevant studies of the brittle fracture criterion here.

We already know that brittle materials do not have a yielding process in tension, i.e. they do not undergo plasticization, but fracture directly in the elastic phase. Therefore, in the case of brittle materials, plastic strain cannot be used as a failure criterion. Since brittle material always fails due to maximum principal stress, the value of Von. Mises equivalent stress will be very different from the maximum principal stress according to the Mohr circle theory, so here again the equivalent stress cannot be used as failure criterion.

In the brittle case, only the maximum principal stress and the elastic strain (or total strain, in this case has the same value as the elastic strain) can be used as fracture criteria, and theoretically both will give the same result. In this paper, the maximum principal stress is used, and the fracture strength is set to 240 MPa according to the material properties.

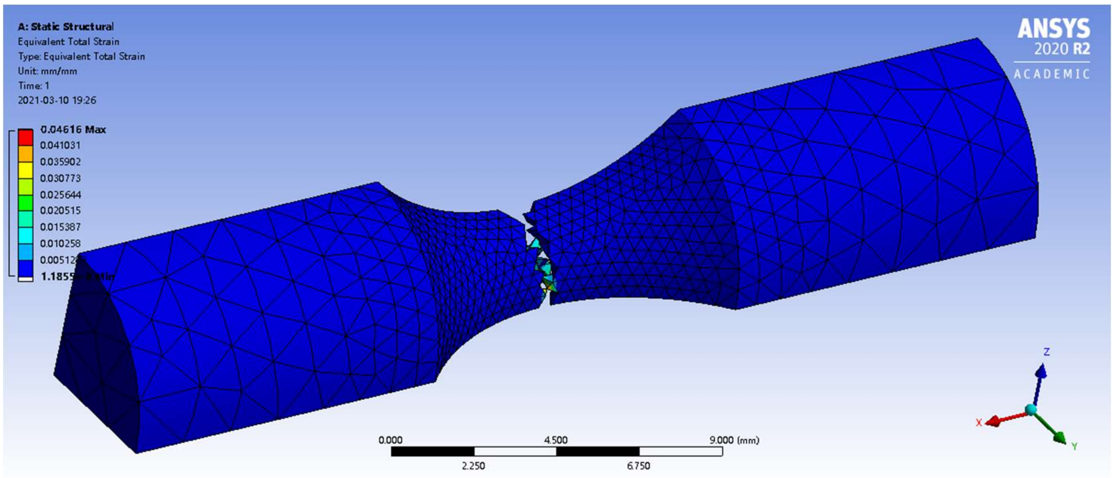
5. APDL command.

The APDL command used for the brittleness simulation has the same cycle logic as the ductility case, so the general template is not changed but only the part of the script about the fracture criteria. If we change “s,eqv” to “s,1”, the logic of the script becomes: if there is an element in the whole specimen that is subjected to a maximum principal stress greater than or equal to 240 MPa, then the element is killed.

6. Debugging.

As with the ductility simulation, the exact amount of stretch at the moment of tensile fracture must be found before activating the APDL command. Since brittle materials are pulled off in the elastic phase, the tensile amount is usually very small and is set to 0.1 mm here. Again, set the substeps in Analysis Settings to 50, 50, 1000 to perform the calculation. In the calculation result, find the time point when the maximum principal stress is just over 240MPa, and follow the path to find the total deformation of the specimen at the corresponding time point, then the stretching amount at the time of fracture is around this value.

Activate the APDL command and change the displacement at the stretching end of the boundary condition to the tensile amount found in the previous section to perform the operation. If the result does not converge, the displacement is adjusted downward appropriately, and if the specimen is not broken, the displacement is adjusted upward appropriately. In this paper, a quite satisfactory result can be obtained when the displacement is 0.0246 mm.

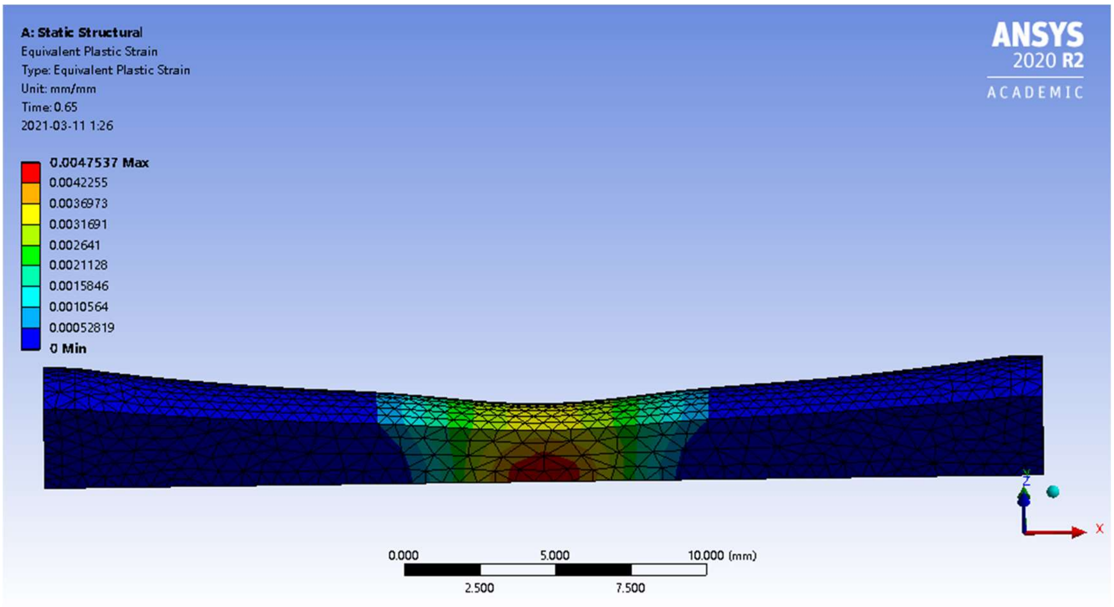


BRITTLE FAILURE OF TENSILE TEST

Failure analysis

Ductile material

1. Necking stage.

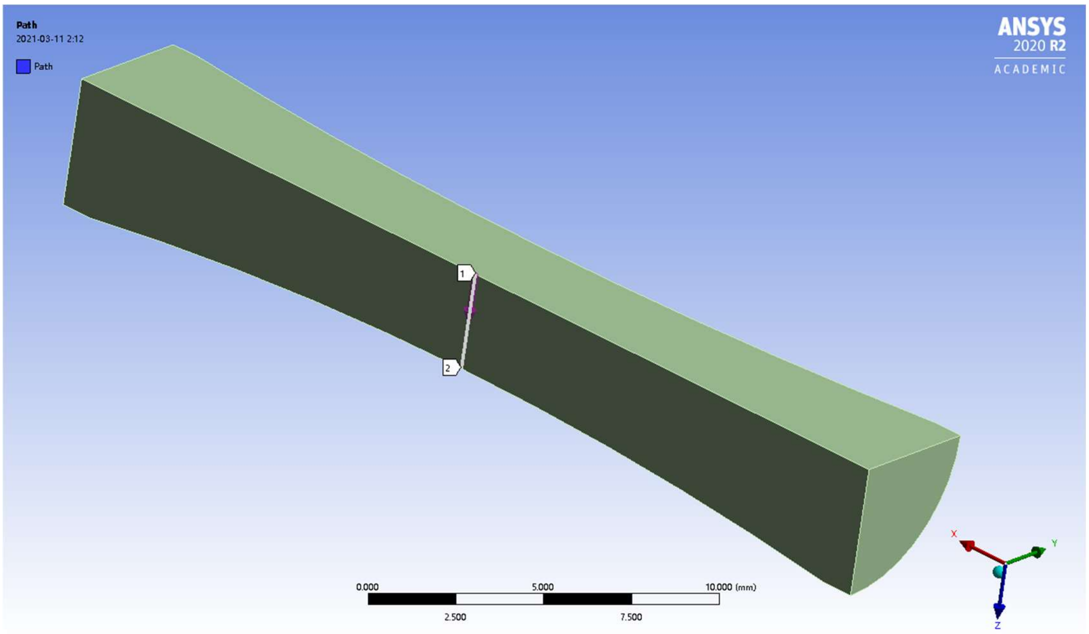


NECKING AT 0.32 MM – 60 SCALE

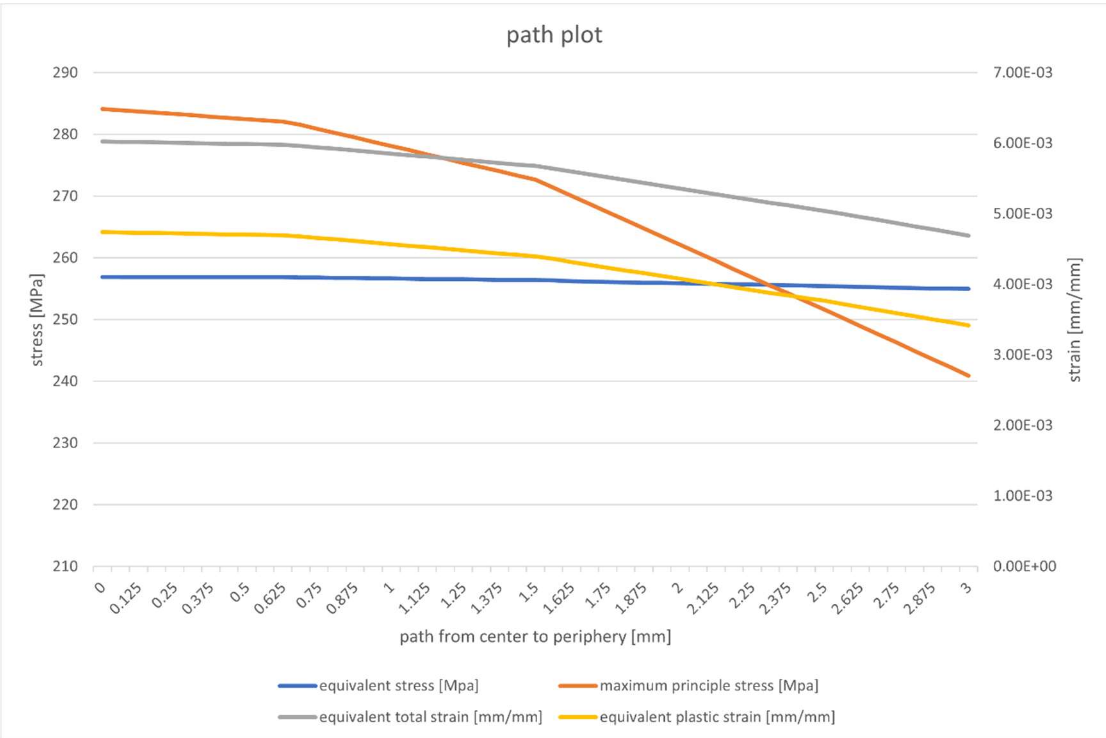
It can be seen that as the stretching proceeds, the necking phenomenon occurs first for the structural steel, which is not obvious and is shown in ANSYS at 60 times the exaggerated size.

It should be explained that the material used in this example is bilinear material, whose σ-ε curve does not have an ultimate yield point compared with the real experimental data. in fact, even for multilinear material, in the implicit algorithm of ANSYS, when the specimen is subjected to a stress exceeding the set maximum real stress, the specimen will still continue to stretch at a fixed slope [9] without the CD segment as shown in FIG 3. However, in order to control the location of the fracture point, the geometry of the specimen is treated in this paper. As it can be seen in the figure, the place where the maximum strain is generated is the interior of the specimen at the smallest cross-section, and the strain decreases gradually from the interior to the periphery. Since the inner parts enters the plastic stage first, while the periphery continues to stretch, the necking phenomenon can still be generated locally. The reason for this relatively small phenomenon is that the fracture criterion set in this case is small and therefore the amount of stretching is small. Without activating the APDL command, if the stretching amount is set to 1mm, the necking phenomenon becomes very obvious.

2. Path plot.

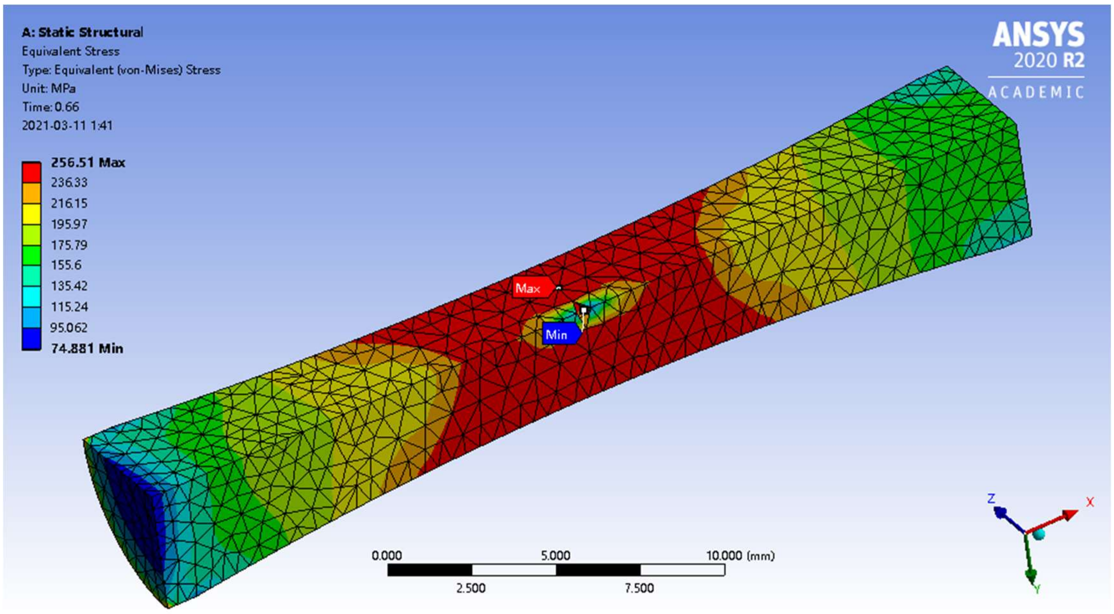


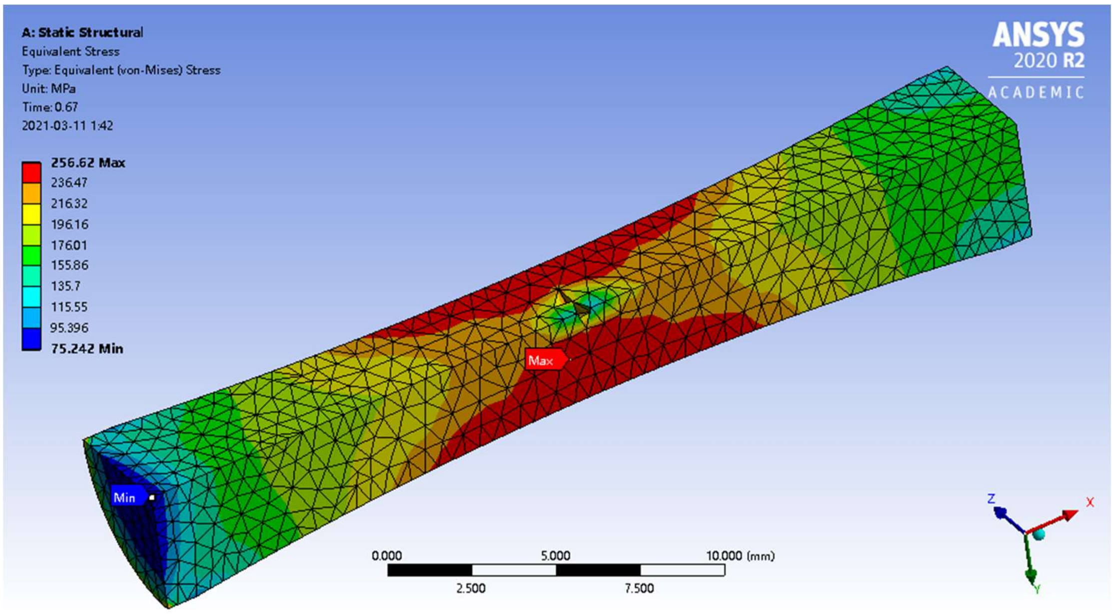
CONSTRUCTION OF THE PATH

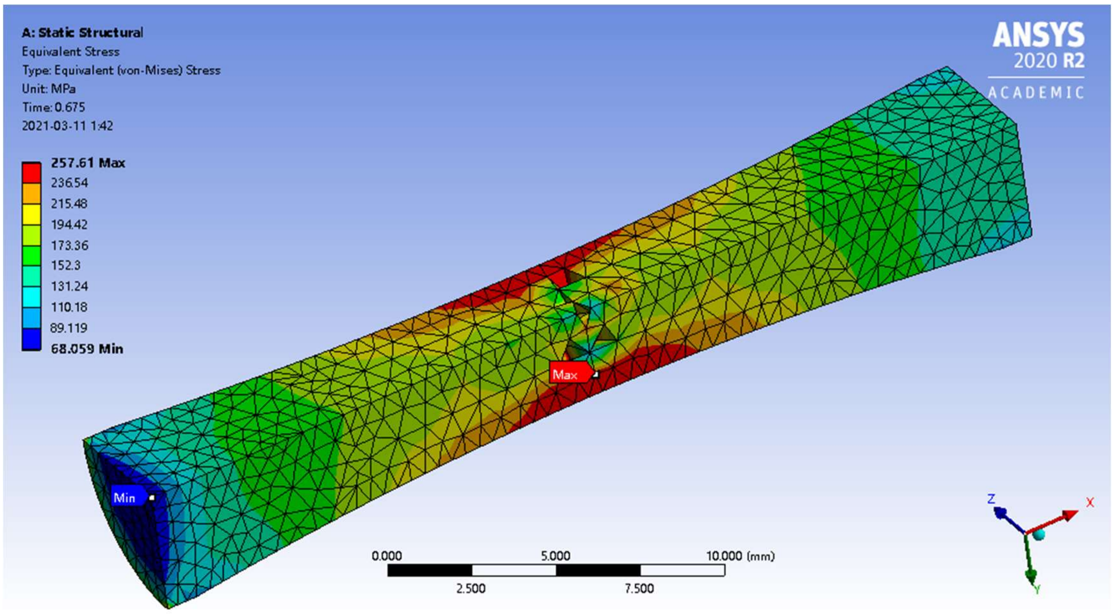


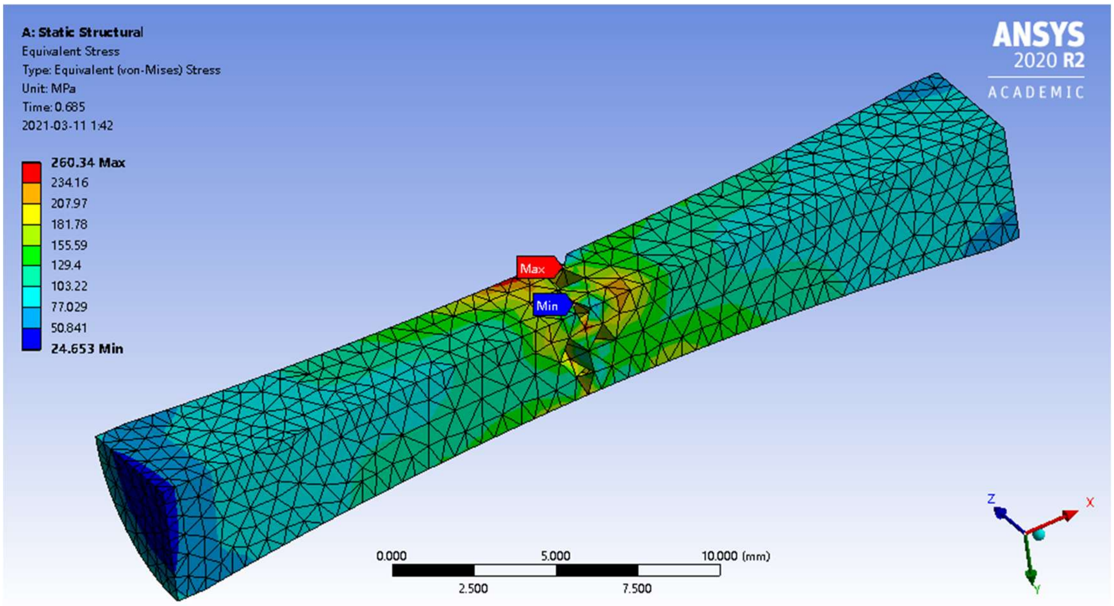
The figure shows a path built at the cross-section of the specimen with the stresses, strains and displacements applied to the points on the path at the instant before the specimen failure. From this diagram, it is also clear that ductile fracture proceeds from the center outward for this reason.

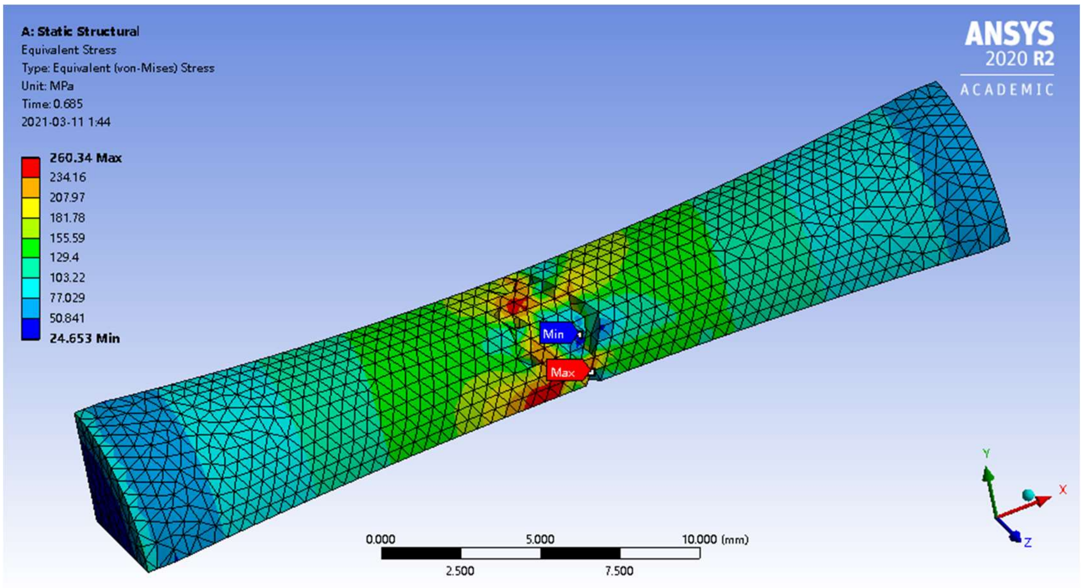
3. Failure process.

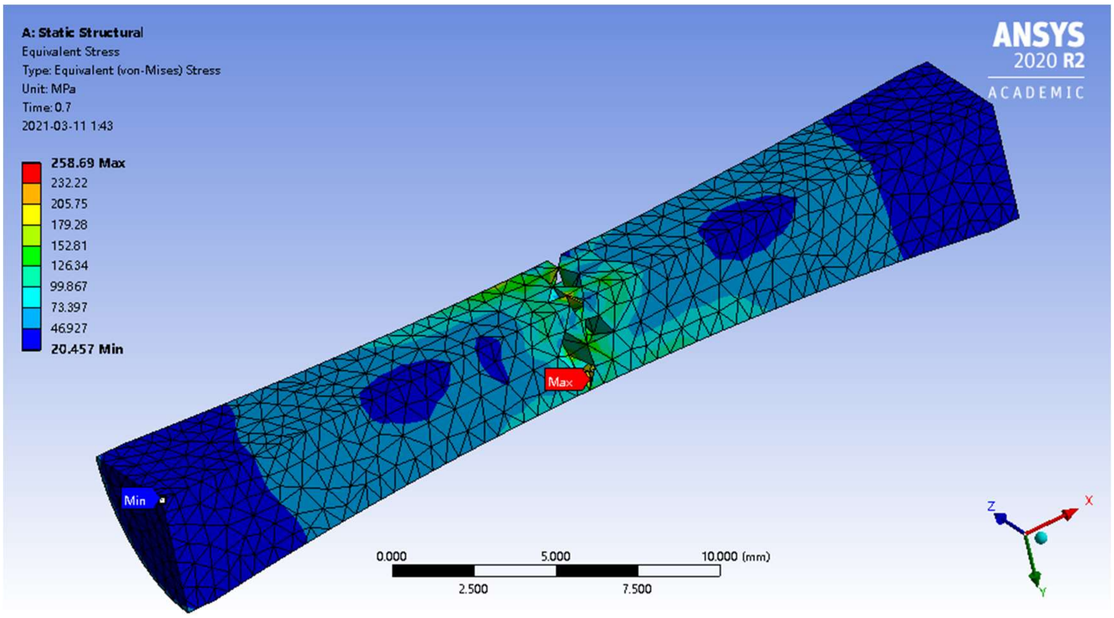


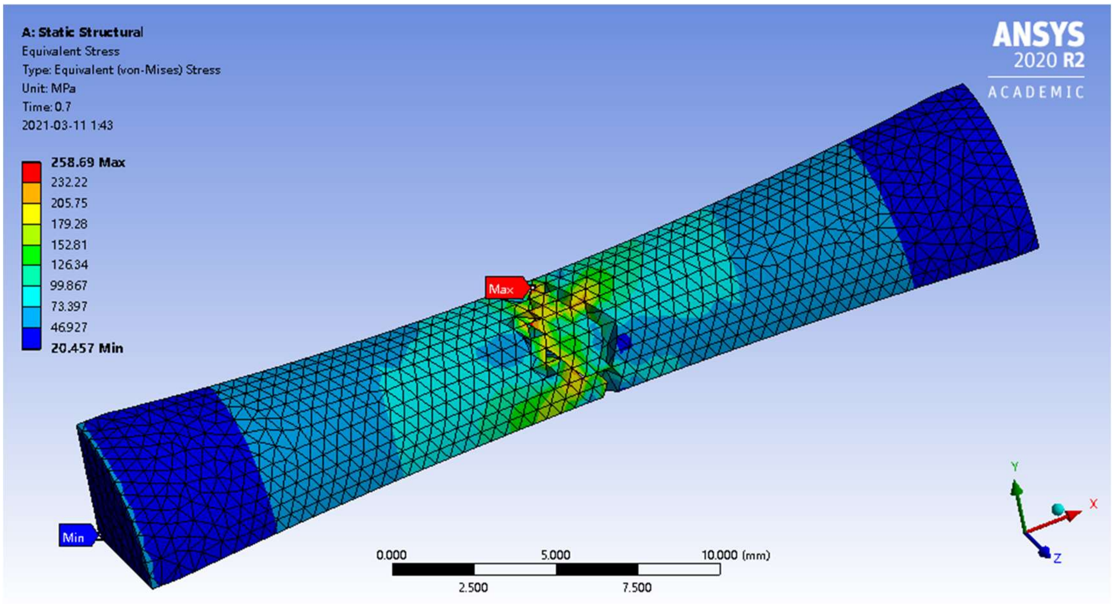



A: Static Structural
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 0.675
2021-03-11 1:42
257.61 Max
236.54
215.48
194.42
173.36
152.3
131.24
110.18
89.119
68.059 Min
ANSYS
2020 R2
ACADEMIC
Max
Min
0.000
5.000
10.000 (mm)
2.500
7.500


A: Static Structural
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 0.685
2021-03-11 1:42
260.34 Max
234.16
207.97
181.78
155.59
129.4
103.22
77.029
50.841
24.653 Min
ANSYS
2020 R2
ACADEMIC
Max
Min
0.000
5.000
10.000 (mm)
2.500
7.500


A: Static Structural
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 0.685
2021-03-11 1:44
260.34 Max
234.16
207.97
181.78
155.59
129.4
103.22
77.029
50.841
24.653 Min
ANSYS
2020 R2
ACADEMIC
Min
Max
0.000
5.000
10.000 (mm)
2.500
7.500

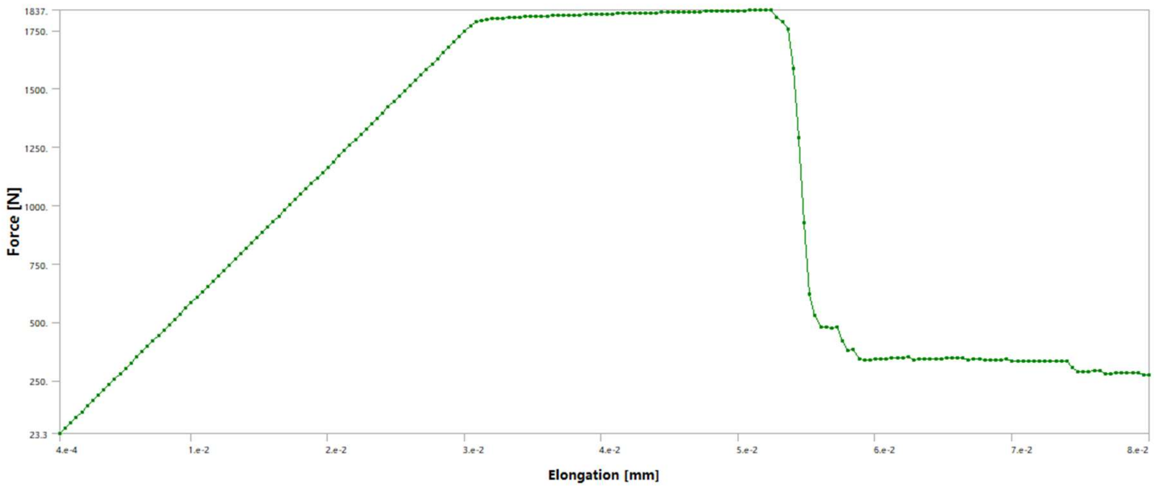




EQUIVALENT STRESS RESULTS AS DISPLACEMENT FROM 0.0528 MM TO 0.056 MM

It can be seen that as the stretching proceeds, the specimen starts to break from the inside and gradually spreads to the periphery, eventually forming a rougher cup-and-cone fracture surface.

4. Force-elongation plot



In order to compare with the real tensile test, a force reaction to the displacement on the outer surface of the tensile end can be calculated, and the force applied to this surface and the corresponding displacement were output and made into a graph as above. The maximum reaction force shown in the graph is 1837 N. Since the failure stress we set is 257 MPa, we can do a quick calculation.

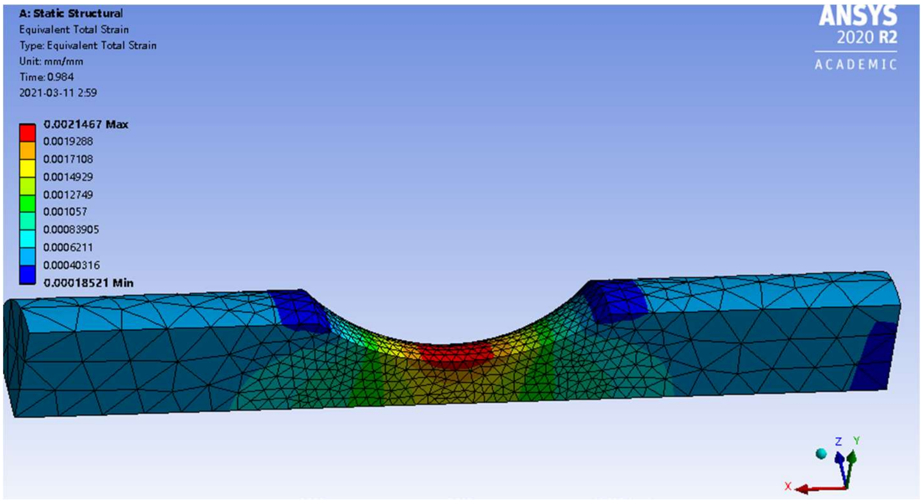
$$F = T \cdot \frac{1}{4} S_0 = 257 \times \frac{1}{4} \times \pi \times 0.003^2 = 1816.63\ [N]$$

It can be seen that this result matches with the real tensile test.

Brittle material

Since the fracture of brittle materials does not undergo yielding and strengthening, but breaks directly in the elastic phase, there is no necking in brittle materials.

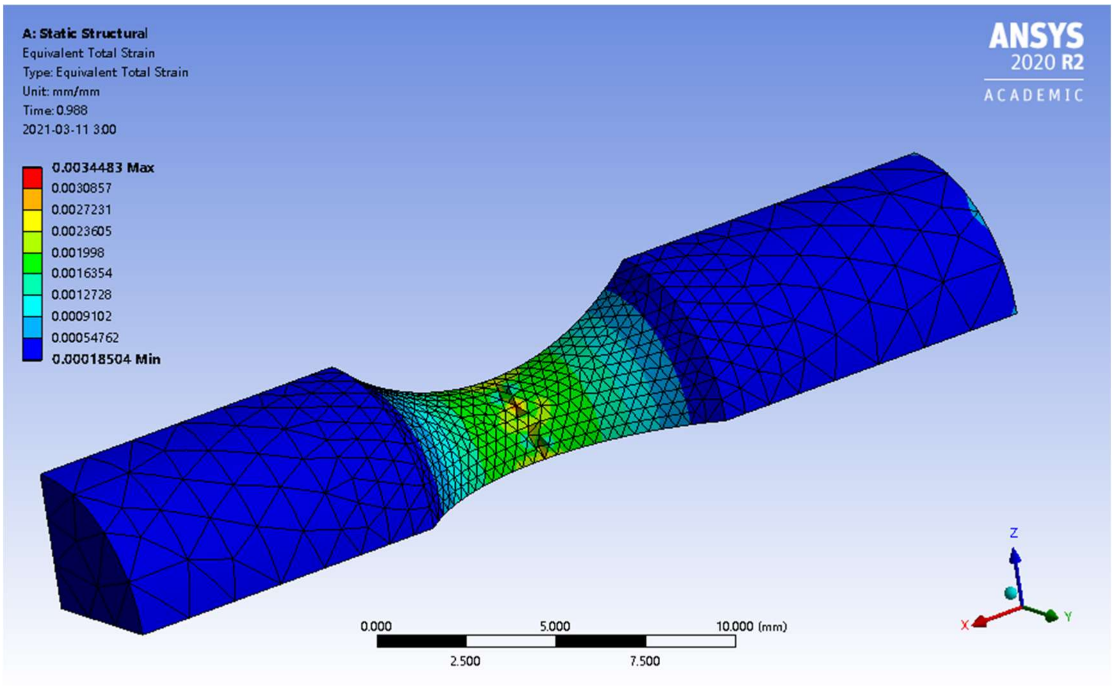
1. Before failure.

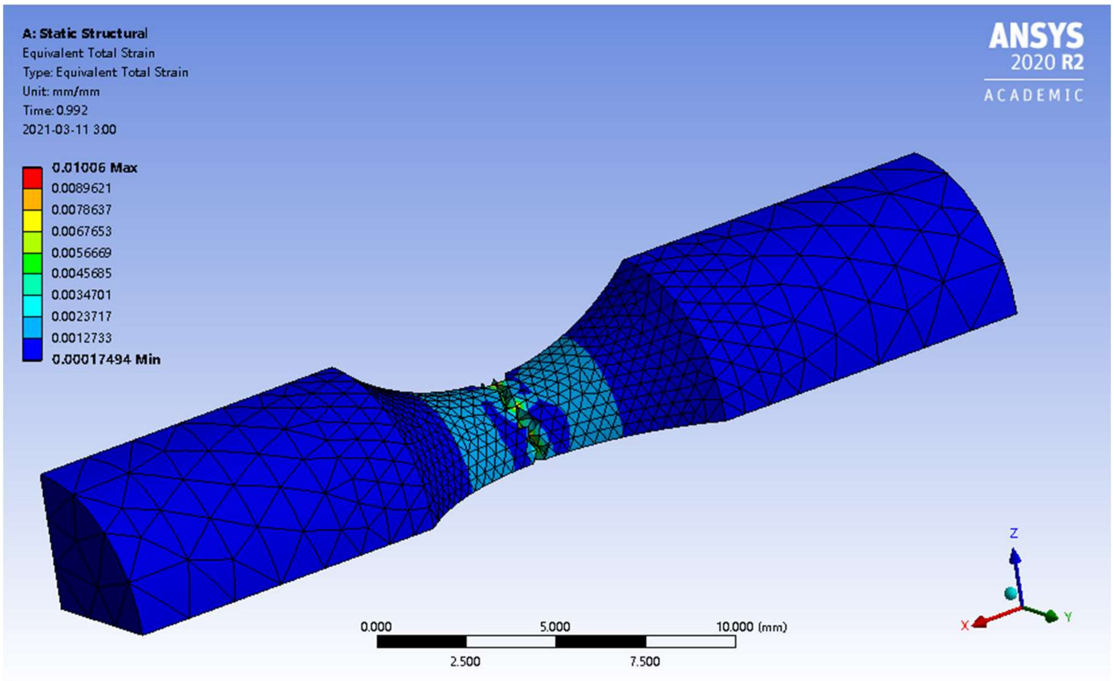


EQUIVALENT STRAIN BEFORE FAILURE

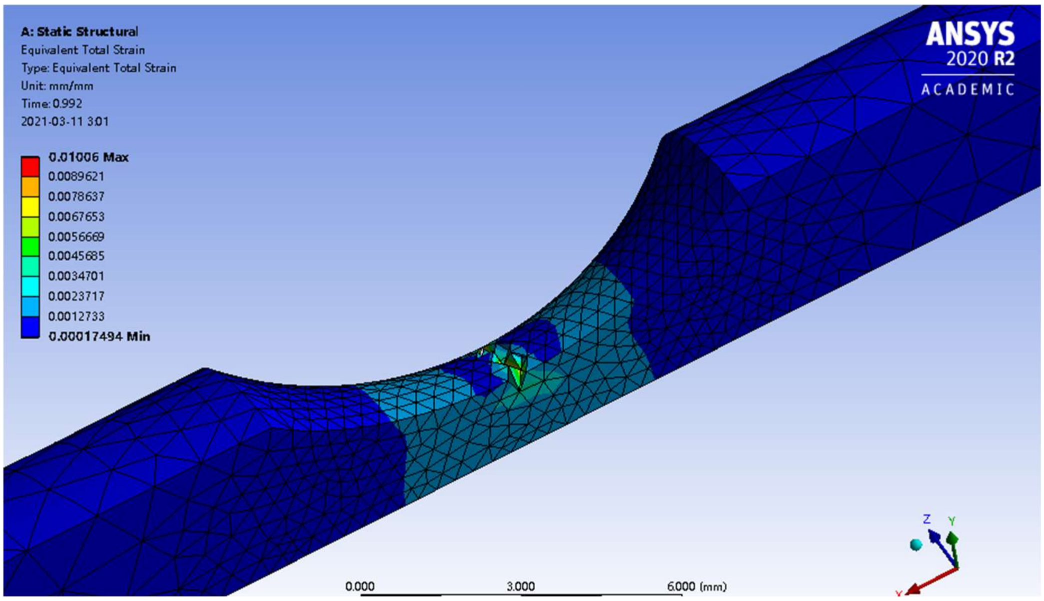
It can be seen that the maximum stress in the brittle material is located at the outermost part of the specimen at the moment before fracture, and there is no concentration phenomenon. This is also consistent with the fact that no necking phenomenon occurs in brittle materials before fracture.

2. Failure process.

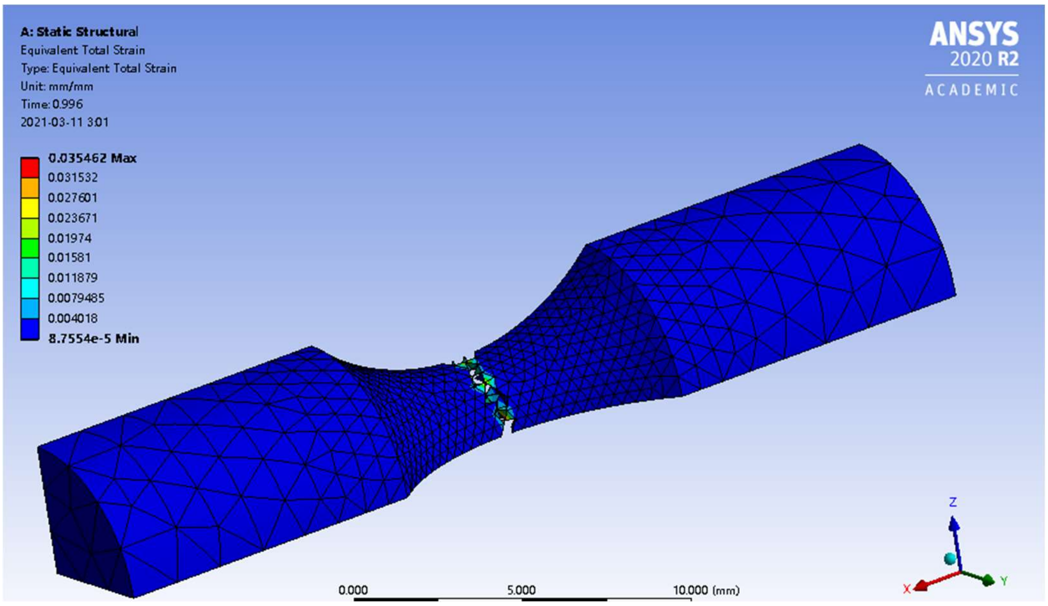


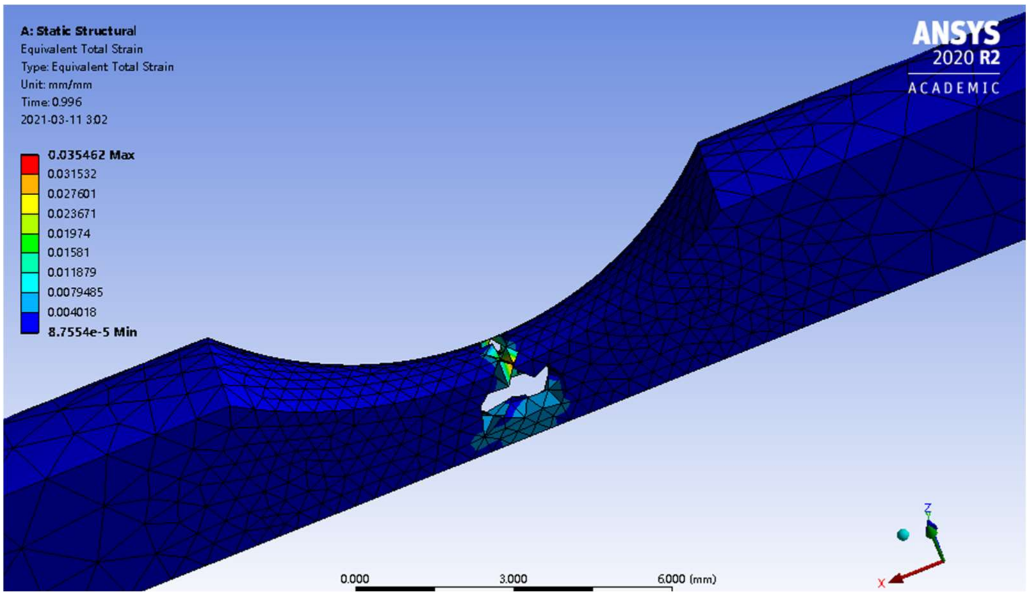


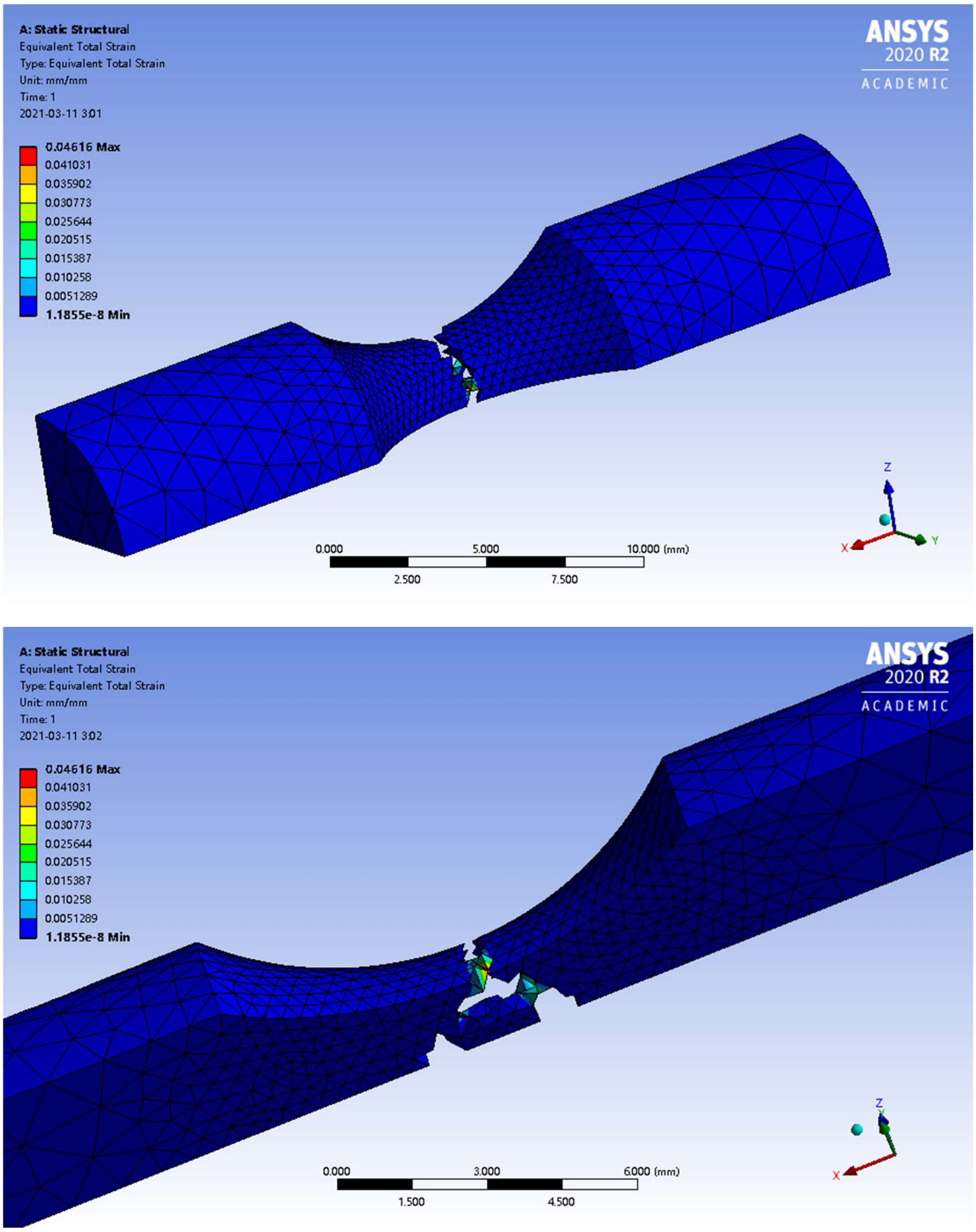
A: Static Structural
Equivalent Total Strain
Type: Equivalent Total Strain
Unit: mm/mm
Time: 0.992
2021-03-11 3:00
0.01006 Max
0.0089621
0.0078637
0.0067653
0.0056669
0.0045685
0.0034701
0.0023717
0.0012733
0.00017494 Min
ANSYS
2020 R2
ACADEMIC
0.000
5.000
10.000 (mm)
2.500
7.500
Z
X
Y



A: Static Structural
Equivalent Total Strain
Type: Equivalent Total Strain
Unit: mm/mm
Time: 0.992
2021-03-11 3:01
0.01006 Max
0.0089621
0.0078637
0.0067653
0.0056669
0.0045685
0.0034701
0.0023717
0.0012733
0.00017494 Min
ANSYS
2020 R2
ACADEMIC
0.000
3.000
6.000 (mm)
Z
Y


A: Static Structural
Equivalent Total Strain
Type: Equivalent Total Strain
Unit: mm/mm
Time: 0.996
2021-03-11 3:01
0.035462 Max
0.031532
0.027601
0.023671
0.01974
0.01581
0.011879
0.0079485
0.004018
8.7554e-5 Min
ANSYS
2020 R2
ACADEMIC
0.000
5.000
10.000 (mm)
Z
X
Y

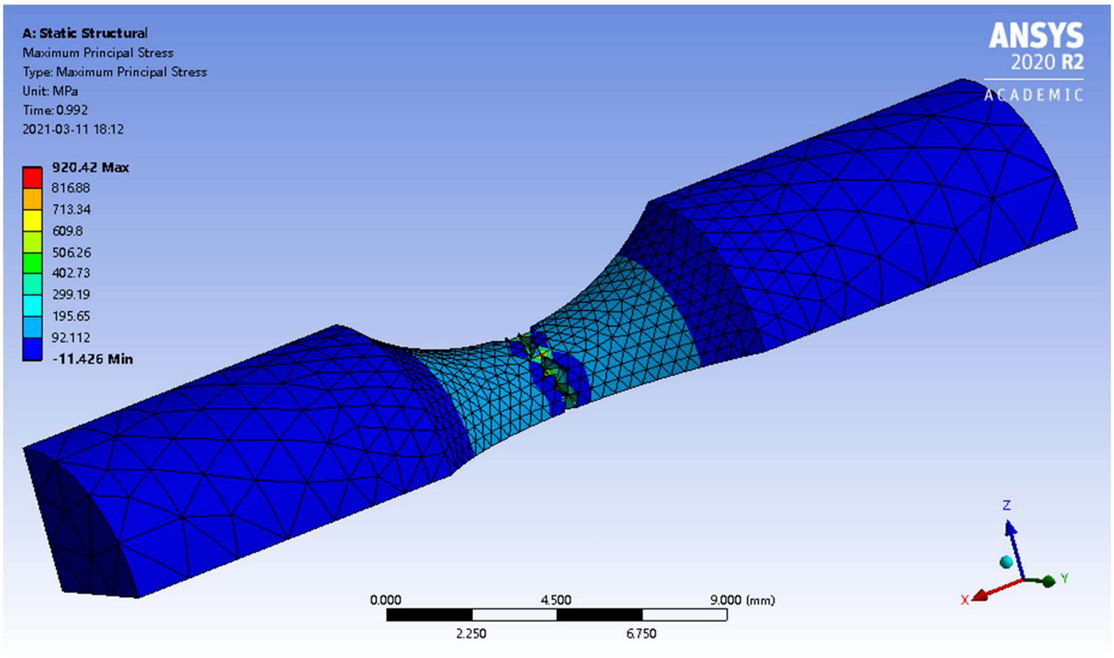

A: Static Structural
Equivalent Total Strain
Type: Equivalent Total Strain
Unit: mm/mm
Time: 0.996
2021-03-11 3:02
0.035462 Max
0.031532
0.027601
0.023671
0.01974
0.01581
0.011879
0.0079485
0.004018
8.7554e-5 Min
ANSYS
2020 R2
ACADEMIC
0.000
3.000
6.000 (mm)
Z
X



EQUIVALENT TOTAL STRAINS AS DISPLACEMENT FROM 0.0243 MM TO 0.0246 MM

It can be seen that the fracture of the brittle material extends from the periphery to the interior as the stretching proceeds. Moreover, the fracture at the periphery is flush. Although a flat fracture surface was not formed due to the mesh problem, the fracture process was consistent with the theory.

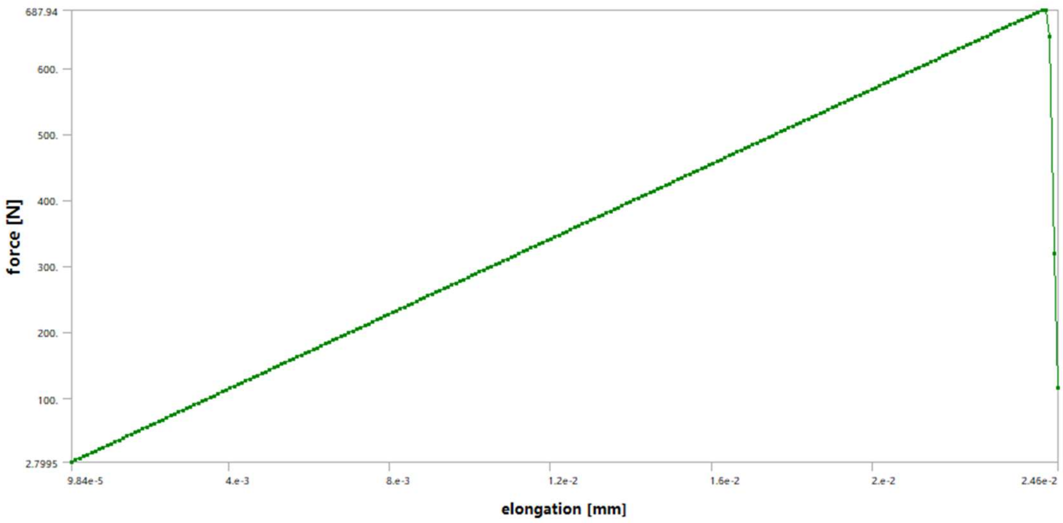
Error analysis:



MAXIMUM PRINCIPAL STRESS IS LARGER THAN REASONABLE RESULT

As shown in the figure above, although the APDL command runs kill the elements in the specimen that are subjected to stresses greater than 240 MPa, the cells cannot be killed completely because the mesh is divided into tetrahedrons, and the residual elements are connected together in the manner shown below. Since the area connected between the tetrahedrons is very small, very large stresses remaining at the elements that not be killed can be generated with little force. Despite this problem in this fracture simulation, the general process is still correct and the results can be verified by the graph of force-elongation.

3. Force-elongation plot.



The above figure can clearly show the whole process of specimen fracture. Before fracture, the force on the tensile end of the specimen is proportional to the amount of stretching. When the stretching amount reaches a certain value, the specimen suddenly fractures and the force on the stretching end disappears rapidly. In order to verify the results, the geometric model of the specimen in this case is calculated by substituting the area of the central section, if the maximum stress on the central section of the specimen at the moment before fracture is 240 MPa, then,

$$F = \sigma \cdot \frac{1}{4} S_0 = 240 \times 10^6 \times \frac{1}{4} \times \pi \times 0.002^2 = 753.98\ N$$

It can be calculated that the maximum force applied to the tensile end is about 754 N. Since the specimen is not simply a column, but has notches, it is clear that there is a stress concentration effect in this case and the factor will be slightly greater than 1.

$$K_t = \frac{\sigma_{max}}{\sigma_{ref}} = \frac{754\ N \times S}{688\ N \times S} = 1.096$$

As the above calculation shows, this result of 688 N is in line with expectations.

Chapter 2: Torsional test

Theoretical model

Torsional problems are a frequent type of problem encountered in engineering. Many products and components are subjected to torsional moments during operation. When engineers attempt to change or optimize the materials used in these products, torsional testing must be performed. For example, the metals used in a vehicle's drivetrain are subjected to complex combinations of loads during mechanical operation, of which torsional moments are a major part. In order to achieve more fuel efficiency, the material of the drive shaft can be changed during the optimization of the vehicle design to reduce the weight of the vehicle. In this case, conducting torsional tests can help engineers find materials that are less dense in mass under conditions that exhibit the required torsional strength to achieve the goal of lighter weight.

The purpose of the torsion test is to determine the behavior of a material under torque or loading conditions that causes shear stresses. The values that can be measured in this test include: the modulus of elasticity in shear, yield shear strength, torsional fatigue life, ductility, ultimate shear strength. Similar data are obtained in tensile tests, which are important in manufacturing to simulate conditions of use, to control and optimize the quality and design of products, and to ensure proper production.

In the laboratory, torque is applied to a metal material specimen (mild steel or cast iron) at room temperature, and the torque and its corresponding torsion angle (generally twisted to fracture) are measured to determine the torsional mechanical properties index of some materials. Like the tensile test, the torsion tester also has two clamps on the same axis, which are used to fix the two ends of the column specimen so that it does not produce displacement other than rotation about the axis. The difference is that the output value is a torque rather than a force.

Ductile material

Mild steel, for example, to do the torsion test in laboratory, the test rig must be applied to opposite torsional moments at the ends of the specimen. Specimens under the action of the external moment, all points in the circular cross-section undergo a pure shear stress state thanks to the features of the torsion phenomenon. With the increase of the external torsional moment, when a certain value is reached, the indicator of the output machine will appear to pause, then this value of the external torsional moment on the indicator is the yield moment M_{es}.

After the yield torsional moment M_{es} is measured, the external torque is continued to be increased on the specimen until the test piece is twisted. At this point, the value of the external torque indicated by the output machine or computer is the maximum torque M_{eb}.

Mild steel specimen in the process of torsional deformation, using the output machine or computer to plot the $M_e - \varphi$ diagram as shown in FIG 4.

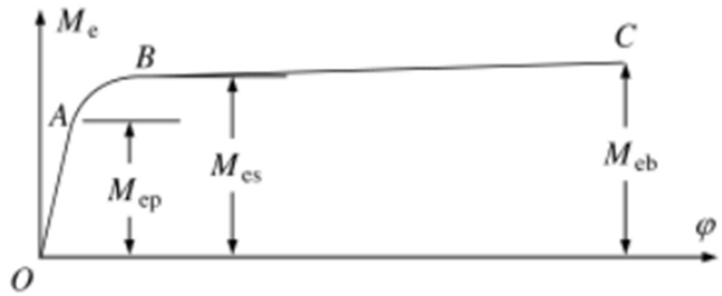


FIG 4 - TORSION DIAGRAM OF LOW CARBON STEEL

In the OA section, the material is in the elastic stage, and the torsional moment and rotation angle of the specimen are proportional to each other. When the moment is removed, the specimen can return to its original state. When the point A in the figure is reached, the proportional relationship between M_e and φ starts to break down. At this point, the shear stress on the surface of the specimen reaches the torsional yield stress τ_s of the material, and the material enters the yielding stage.

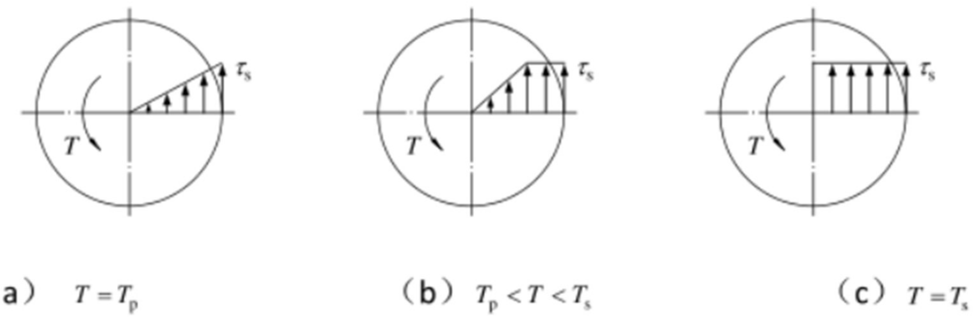


FIG 5 - DISTRIBUTION OF TANGENTIAL STRESSES IN THE CROSS-SECTION OF A CYLINDRICAL SPECIMEN OF DUCTILE MATERIAL DURING TORSION

If the corresponding external torsional moment M_{ep} can be measured at this time, the torsional yield stress can be obtained after a series of calculation. As shown in FIG 5 (a), it corresponds to the stage at point A in FIG 4, then the yield torque T_p can be calculated by:

$$T_p = W_p \cdot \tau_s$$

Where,

$$W_p = \frac{\pi d^3}{16}$$

Is the torsional cross-sectional coefficient of the specimen within the standard distance.

Since

$$T_p = M_{ep}$$

Then

$$\tau_s = \frac{M_{ep}}{W_p}$$

After passing point A, a ring-like plastic region appears in the cross-section of the specimen, as shown in FIG 5 (b). The peripheral part of the specimen cross-section enters the plastic stage, while the inner ring is still in the elastic stage. If the plasticity of the material is good, and when the plastic region is extended close to the center, the shear stress at the points on the periphery of the cross-section still does not exceed the torsional yield stress τ_s, the distribution of shear stress at this time can be simplified to the case shown in FIG 5 (c), and the corresponding torque T_s is

$$T_s = \int_0^{d/2} \tau_s \rho 2\pi\rho d\rho = 2\pi\tau_s \int_0^{d/2} \rho^2 d\rho = \frac{\pi d^3}{12}\tau_s = \frac{4}{3}W_p\tau_s$$

Since

$$T_s = M_{es}$$

We can easily have

$$\tau_s = \frac{3}{4}\frac{M_{es}}{W_p}$$

Whether from the indicator of the output machine or from the curve drawn by the computer, the position of point A is not easy to determine precisely, while the position of point B is more obvious. Of course, this calculation method also has defects, it is correct only when the actual stress distribution is completely in accordance with FIG 5 (c). For less plastic materials, the torsional shear stress obtained by this calculation method varies greatly. As can be seen from FIG 4, when the external torsional moment exceeds M_{es}, the torsion angle φ increases rapidly, while the external torsional moment M_e increases very little, and BC approximates a straight line. Therefore, it can be assumed that the distribution of tangential stress on the cross section is as shown in FIG 5 (c), except that the value of tangential stress is larger than τ_s. According to the measured external torsional moment M_{eb} of the specimen at fracture, the torsional strength can be obtained as

$$\tau_b = \frac{3}{4}\frac{M_{eb}}{W_p}$$

As can be seen in the laboratory, the fracture of the mild steel specimen is perpendicular to the axis, indicating that the damage was caused by tangential stress. A laboratory photograph of a mild steel twist fracture is shown below:

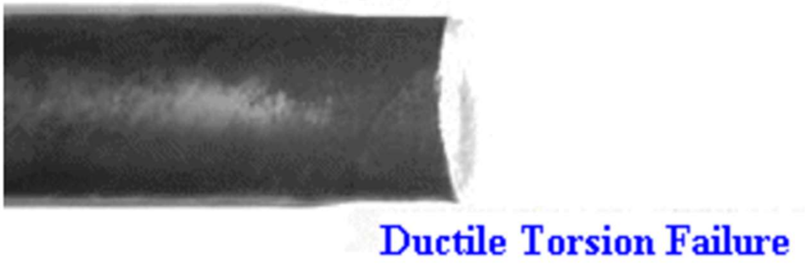


Photo by Jeff Thomas, November 1997

Brittle material

Taking gray cast iron as an example, the torsional strength properties of brittle materials are similar to the tensile strength properties in that they both fracture in the elastic phase. Therefore, for gray cast iron specimens, only the maximum external moment M_{eb} (in the same way as for ductile materials) to which they are subjected needs to be measured in the laboratory, and the torsional strength is

$$\tau_b = \frac{M_{eb}}{W_p}$$

The fracture of the gray cast iron specimen in the laboratory is at an angle of approximately 45° to the axis along the helix direction, indicating that the damage is caused by tensile stress. A laboratory photograph of torsional fracture of gray cast iron is shown in the following figure:

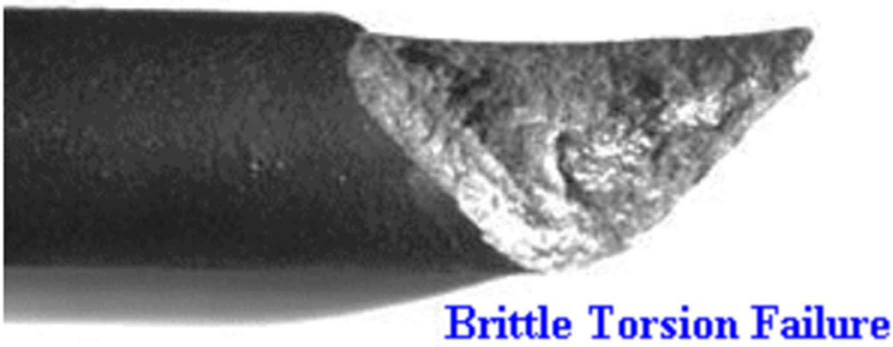


PHOTO BY JEFF THOMAS, NOVEMBER 1997

Building model

For fracture simulation of ductile material

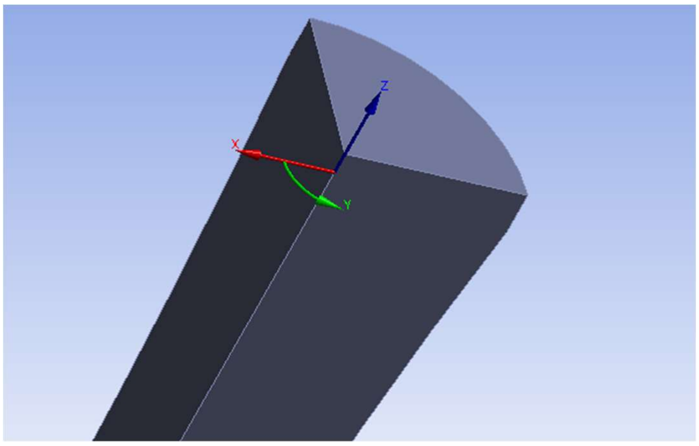
1. **Material.**

   Same as the tensile test simulation, the torsional test simulation for ductile materials was performed using the structural steel NL provided by ANSYS and adding the corresponding isotropic elasticity data. Values are the same as in TABLE 1.

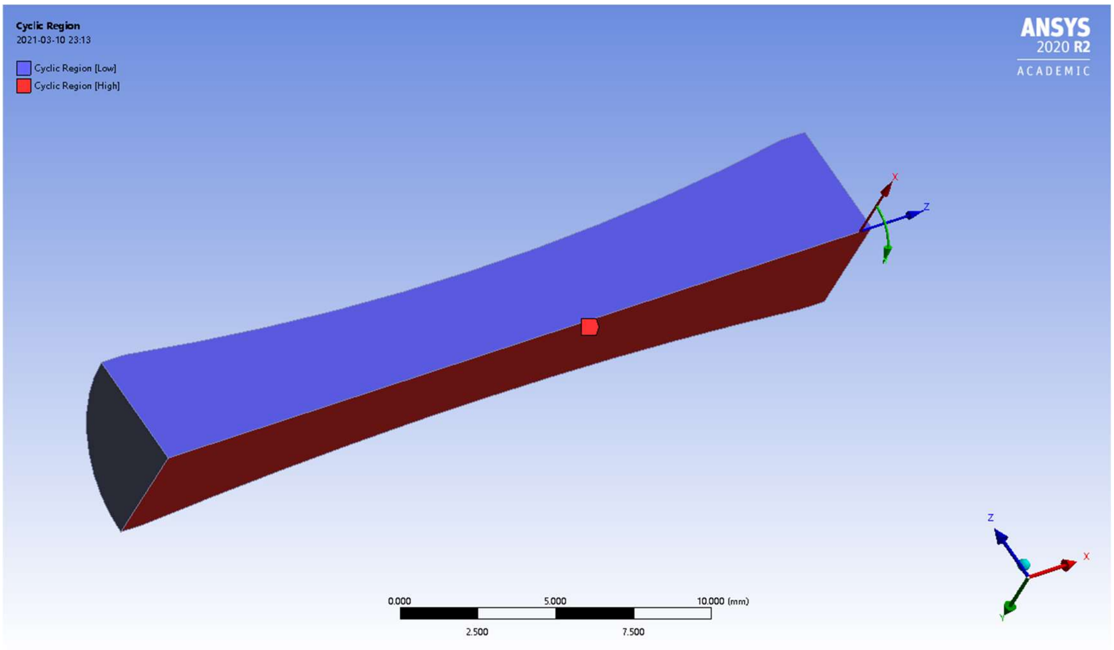
2. **Geometry.**

   The geometry in this case is set to be the same as the ductile case in tensile test simulation.

   In the torsional test, elements on the specimen are displaced in both radial and tangential directions, so the symmetry can no longer be simulated with the frictionless support used in the tensile test. In this case, the cyclic region in the symmetric tool provided by ANSYS Workbench is used to simulate the symmetry. As shown in picture named as "Cylindrical Coordinate System" below, a new cylindrical coordinate system is created, and the blue section is set as the lower face and the red section is set as the upper face in this coordinate system, then ANSYS will simulate the complete symmetric model generated by the rotation of these two sections (as shown in the picture named as "Symmetric Geometry Simulation")



CYLINDRICAL COORDINATE SYSTEM



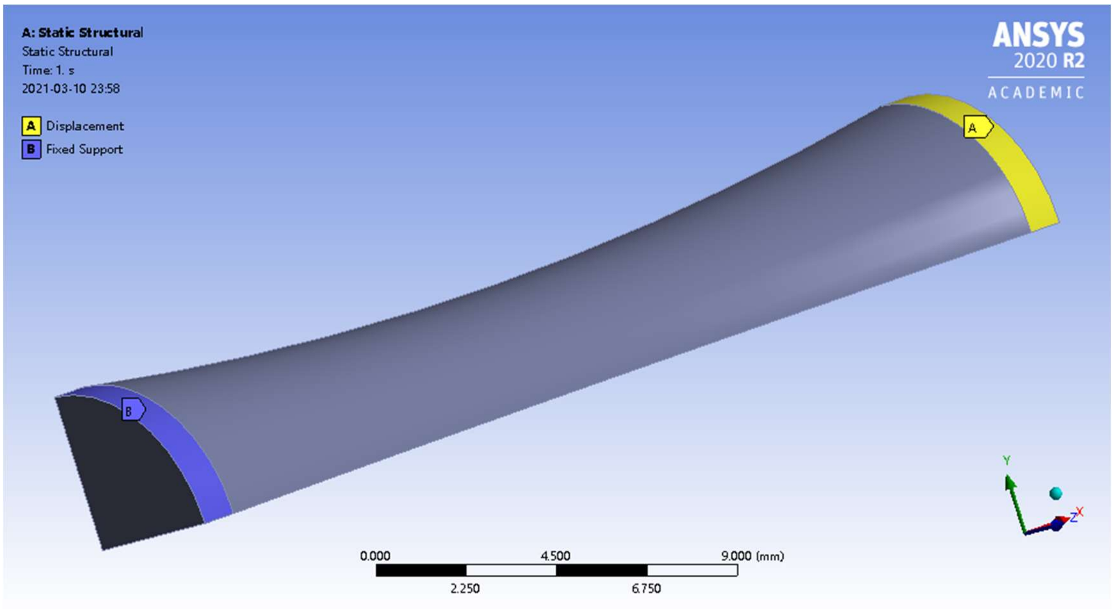
SYMMETRIC GEOMETRY SIMULATION

3. Meshing.

Due to the uniform geometric model structure of this surface, adjusting the regional fineness of the mesh will lead to unconverging results, so a uniform mesh division is still used in this case, and the results are similar to the ductile case in Chapter1.

4. Boundary conditions.

In order to simulate the fixture of the torsion tester, we set fixed support on the surface of one end of the specimen and apply the moment to the other end. There are two ways of applying moments here: applying moment on the edge face and applying a circular displacement based on the cylindrical coordinate system on the peripheral surface of the column. Since APDL command is more applicable to the constraints of displacement and the amount of displacement is easier to control compared to moment, the latter is used in this example, as shown in the figure below (named as Boundary Conditions Settings). To ensure the convergence of the results, more accurate displacements need to be obtained after debugging.



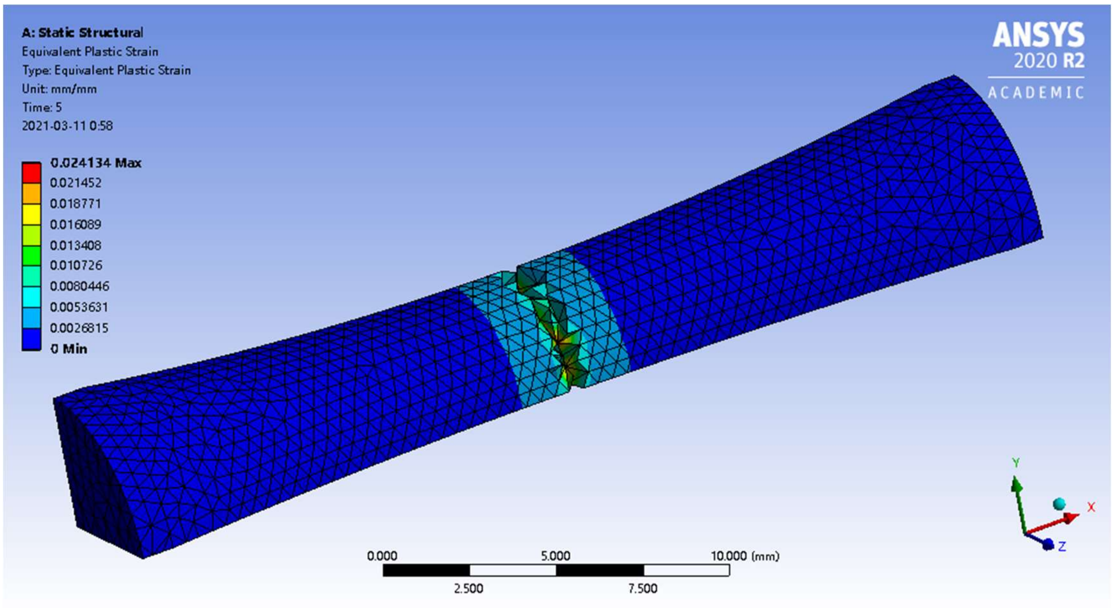
BOUNDARY CONDITIONS SETTINGS

5. **APDL commands.**

In the torsional test simulation, the APDL command used in the previous section is still used because the fracture cycle logic is the same, i.e., the element subjected to a stress greater than or equal to the fracture strength is killed by comparing the stress applied to each element on the specimen with the set fracture strength value. As in the tensile test for ductile materials, in the torsion test, fracture is still judged by the equivalent stress (in APDL command is "s,eqv"). For bilinear material, the fracture strength used in this example is 257 MPa.

6. **Debugging.**

Before activating the APDL command, the exact fracture torsional displacement is obtained using the method mentioned in the previous chapter and this displacement is applied as a boundary condition to the unanchored end surface of the column of the specimen. After activating the APDL command, a more desirable fracture state is obtained by fine-tuning the displacement amount up and down.



DUCTILE FRACTURE OF TORSION TEST

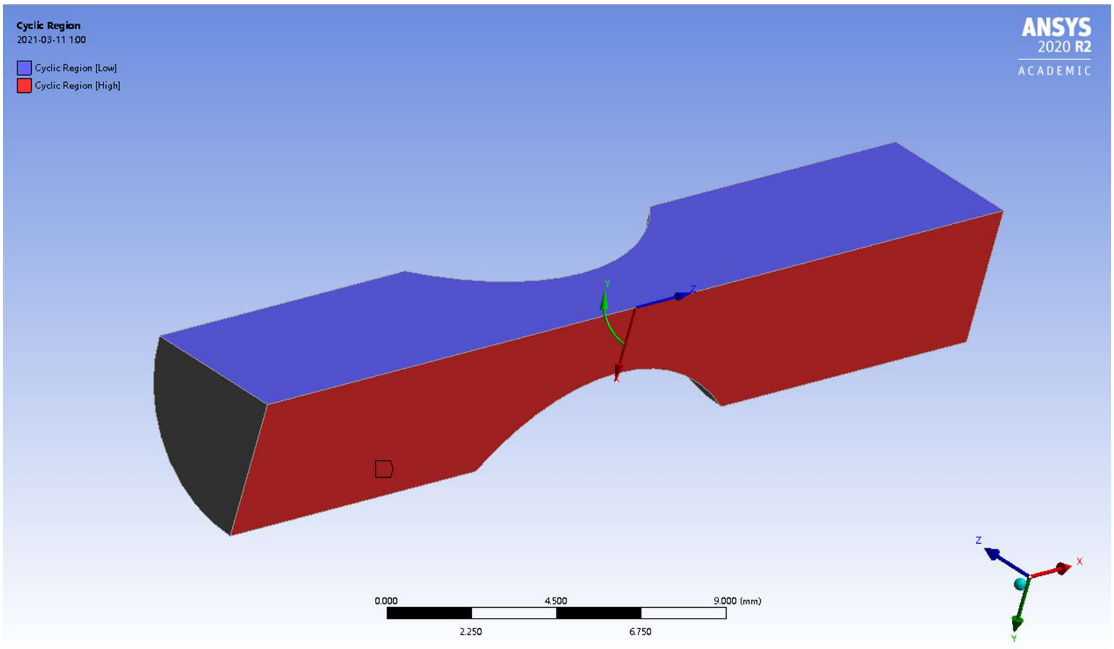
For fracture simulation of brittle material

1. Material.

The brittle material is still selected from the gray cast iron provided by ANSYS. The values are the same as TABLE 2.

2. Geometry.

Similar to the tensile test simulation, the torsion simulation of brittle material also uses hourglass shaped specimen with the same dimensions as the model in Chapter 1. The difference is that the torsional model requires the use of the symmetry tool to simulate the entire specimen, and the procedure is the same as for the ductile case, as shown below for the cyclic region.



SYMMETRIC GEOMETRY SIMULATION

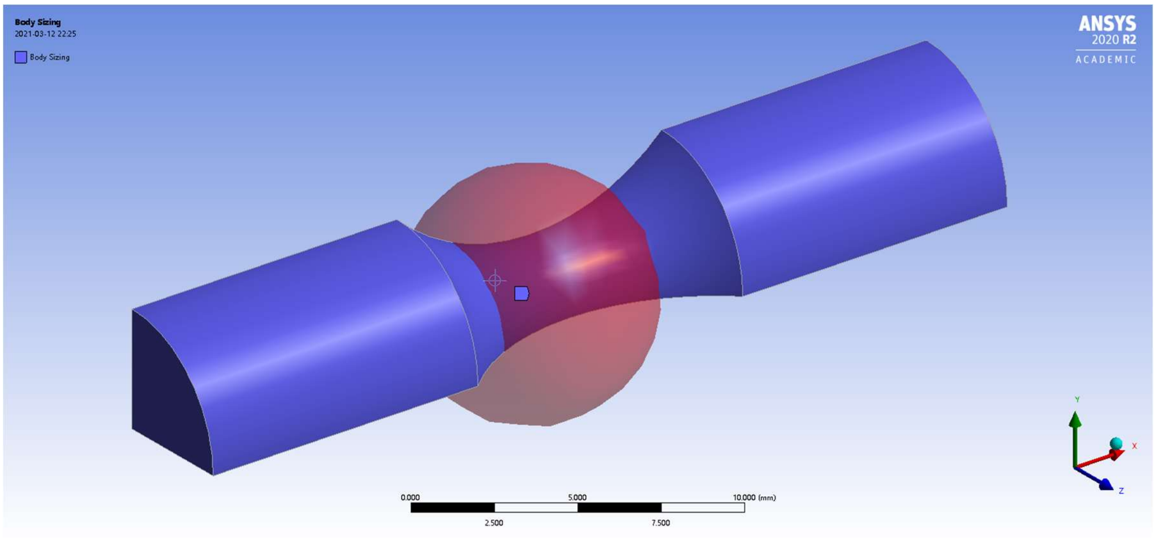
3. Meshing.

Use the sphere of influence type in body sizing to control the fine mesh within the spherical area shown below. Unlike the tensile test simulation, the number of nodes must be adjusted to be smaller in this case because the symmetry tool will increase the computational volume, so the spherical radius in this case is smaller than the tensile case and the growth rate is larger, set to 1.8.

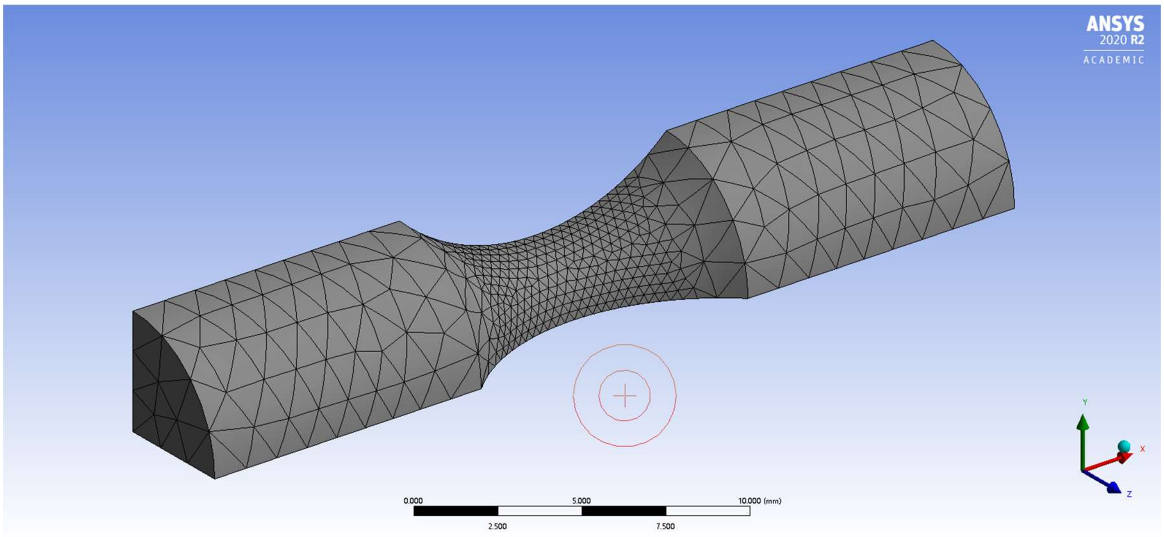
Details of "Body Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Sphere of Influence
Sphere Center	Global Coordinate System
Sphere Radius	4.0 mm
Element Size	0.36 mm

BODY SIZING DATA



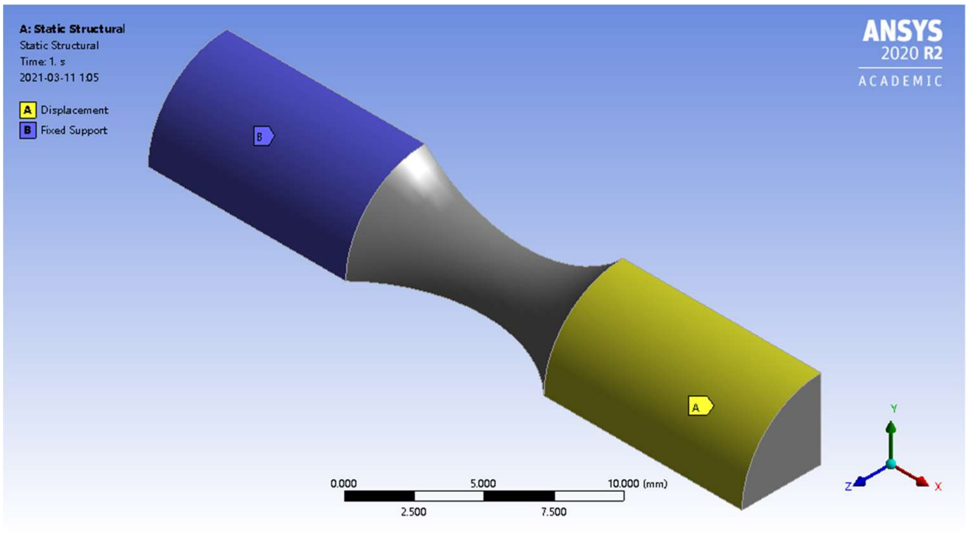
SPHERE OF INFLUENCE



MESHING

4. Boundary conditions.

As in the ductile case, the fixed support and displacement are placed on the outer surface of each end of the specimen, as shown in the figure below.



BOUNDARY CONDITIONS OF BRITTLE TORSION

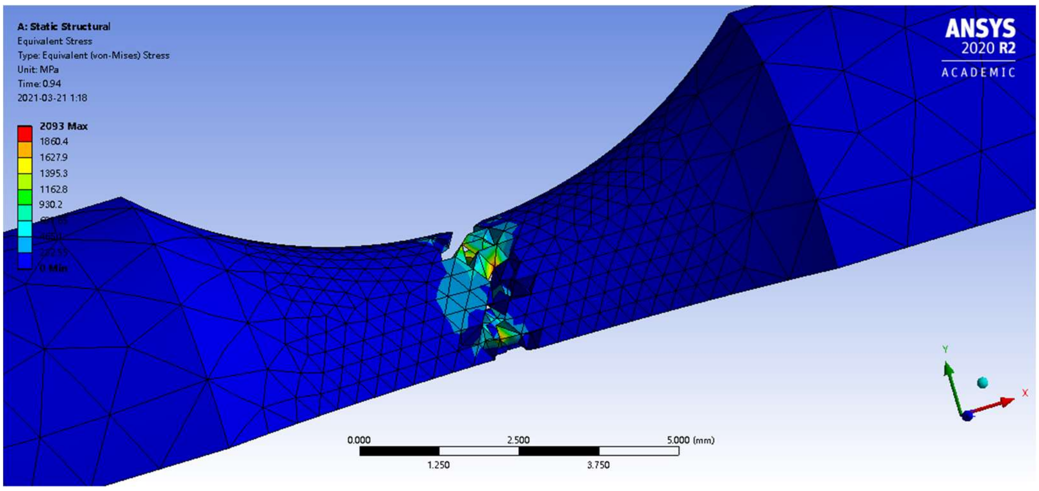
5. APDL command.

The fracture of the brittle material stops only at the elastic stage, when the fracture of the specimen is controlled by the maximum principal stress (in APDL command is “s,1”). The fracture criterion is set to maximum principal stress=240MPa.

6. Debugging.

The commissioning principles and procedures are the same as for the torsional test simulation of ductile materials.

A desirable fracture state can be obtained as follow.

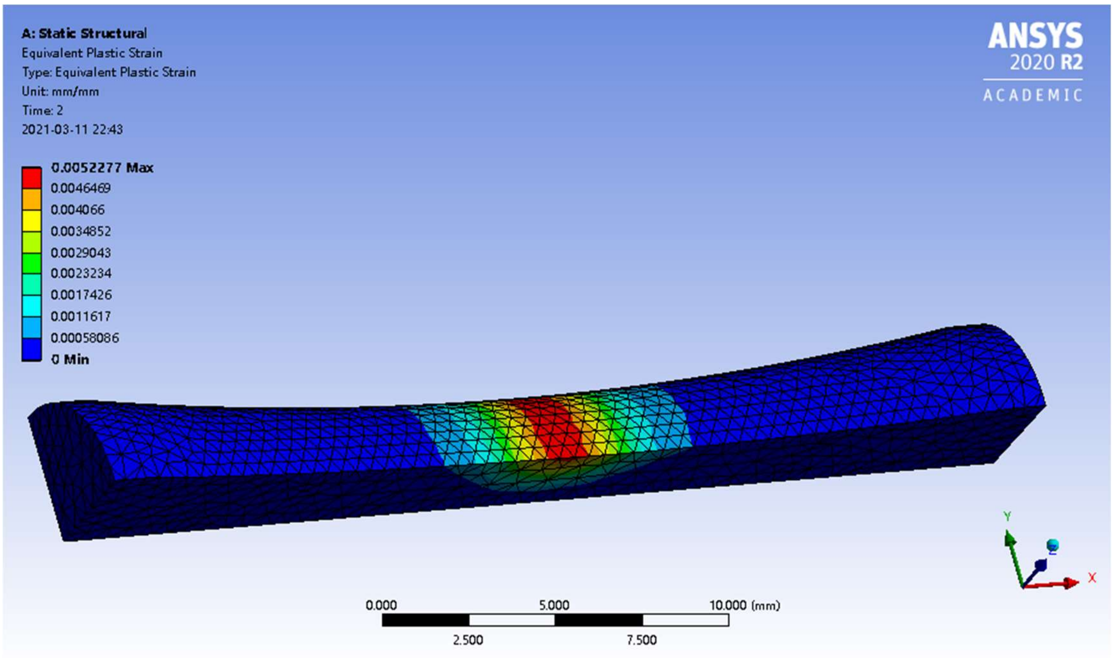


BRITTLE FRACTURE OF TORSION TEST

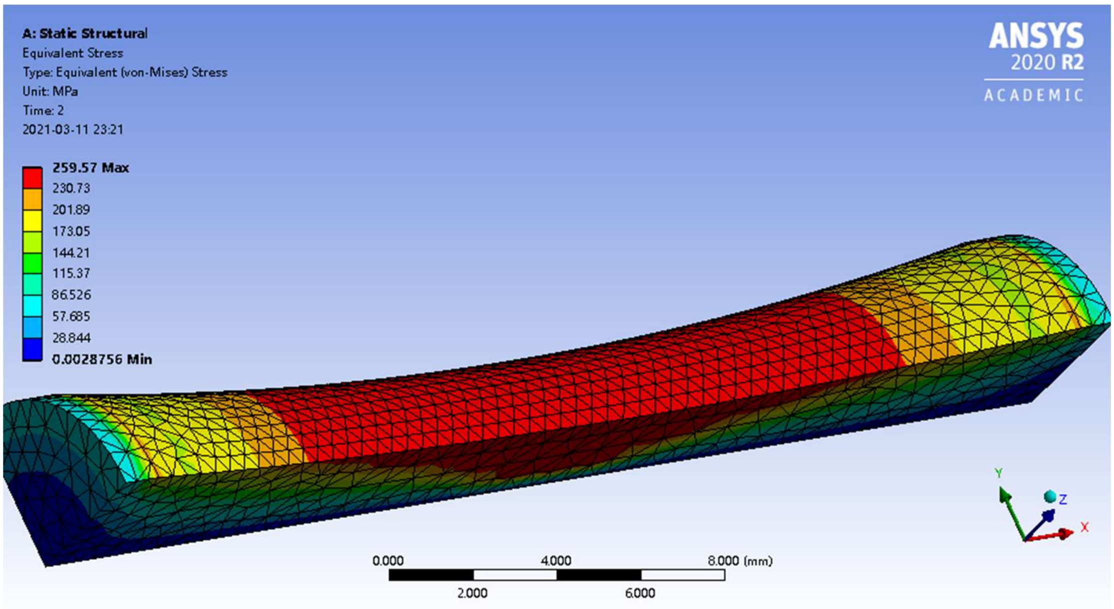
Failure analysis

Ductile material

1. Before failure.



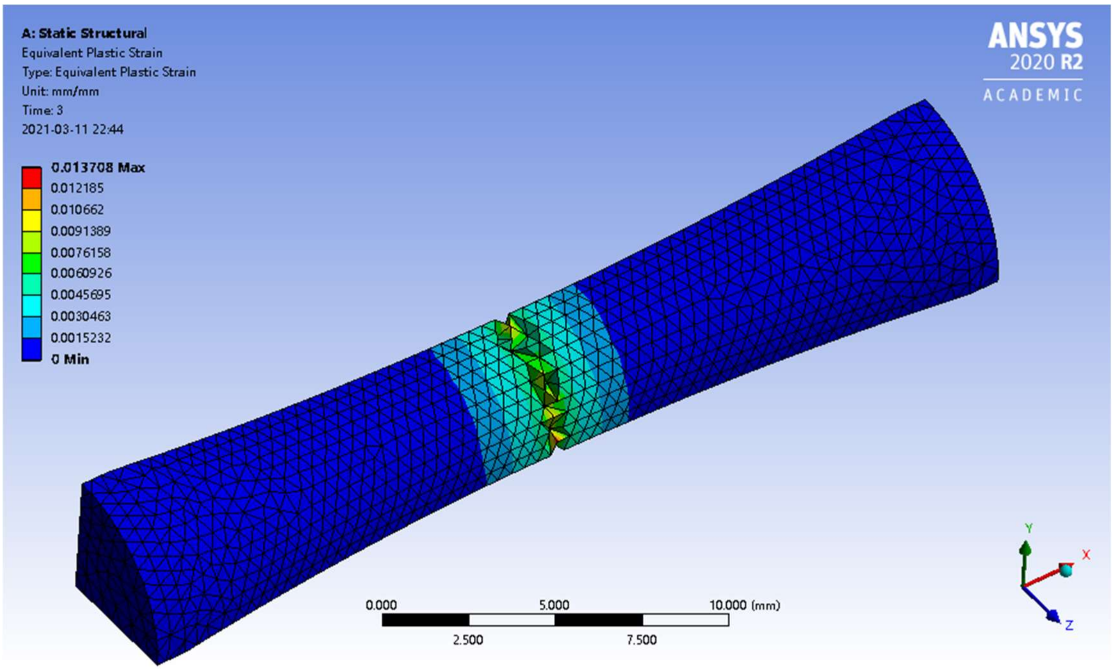
EQUIVALENT PLASTIC STRAIN BEFORE FAILURE

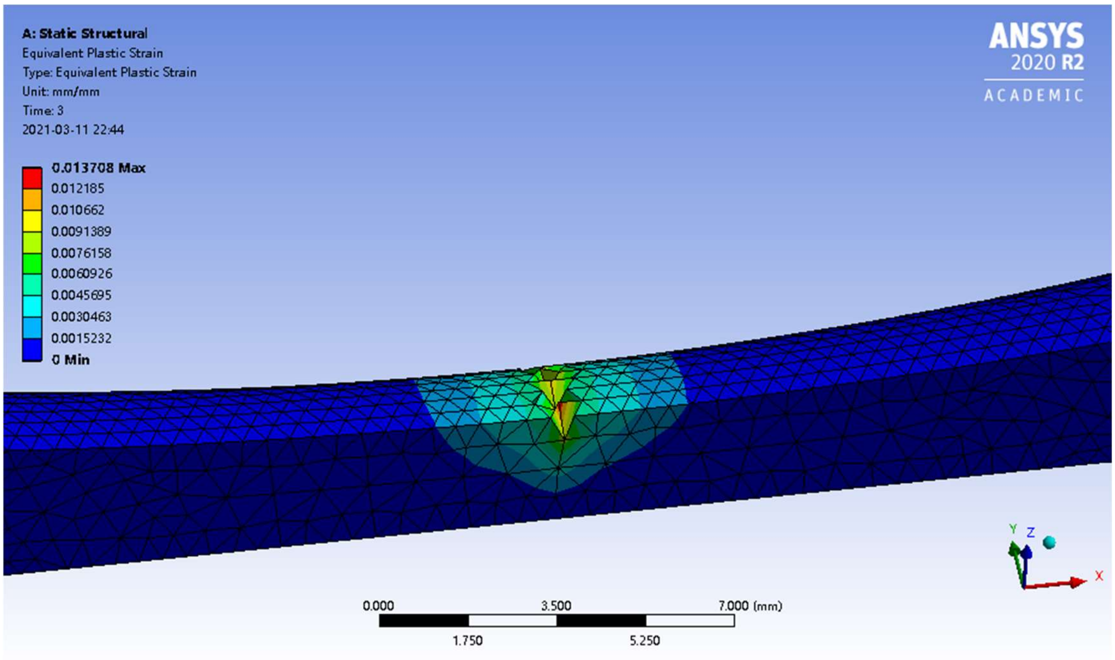


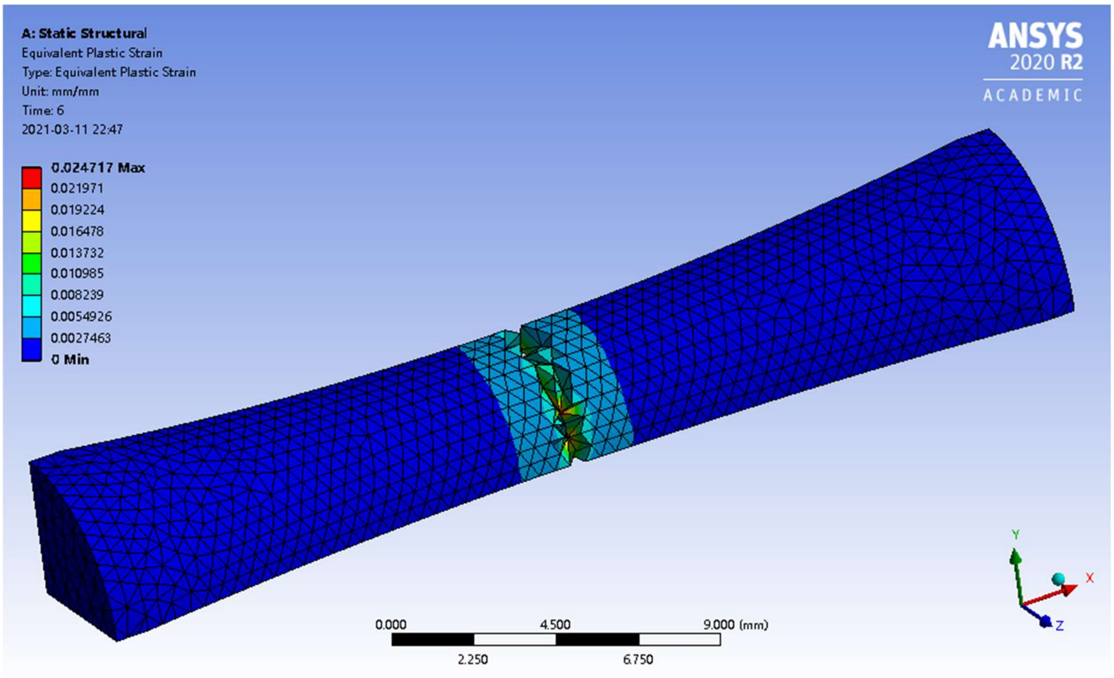
EQUIVALENT STRESS BEFORE FAILURE

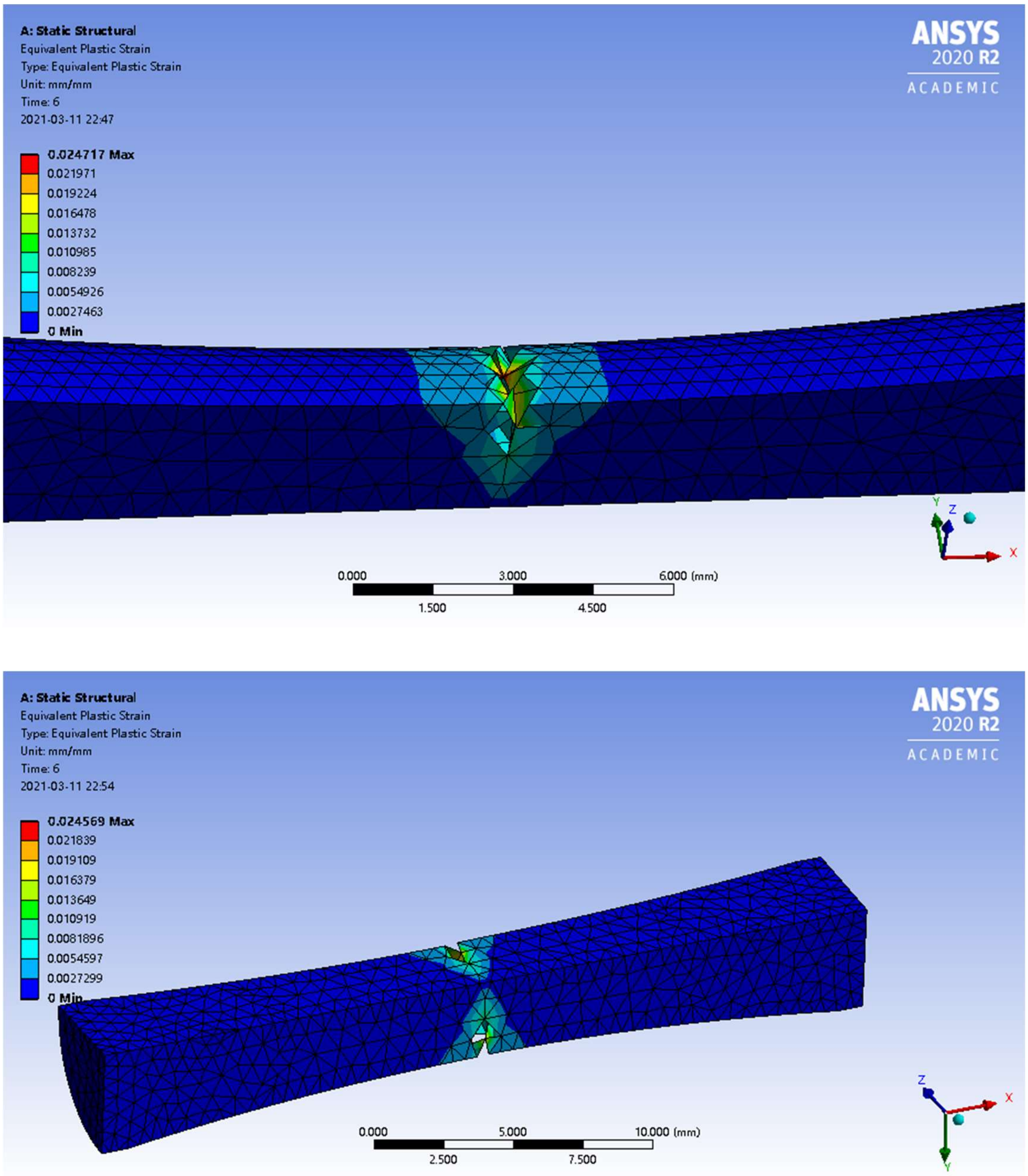
A schematic diagram of the results of the equivalent plastic strain and the equivalent stress at the moment before fracture is shown in the figure. It can be seen that the maximum strain occurs at the outermost part of the specimen and then gradually decreases towards the inner part of the specimen. This result indicates that when the specimen is twisted to this extent, i.e., the moment before fracture, the periphery of the specimen has already undergone a large plastic deformation, while the interior is still in the elastic range. As the theoretical explanation of the torsion test, FIG 5, the specimen is in the state shown in FIG 5 (b).

2. Failure process.




A: Static Structural
Equivalent Plastic Strain
Type: Equivalent Plastic Strain
Unit: mm/mm
Time: 3
2021-03-11 22:44
0.013708 Max
0.012185
0.010662
0.0091389
0.0076158
0.0060926
0.0045695
0.0030463
0.0015232
0 Min
ANSYS
2020 R2
ACADEMIC
0.000
3.500
7.000 (mm)
1.750
5.250
Y
Z
X

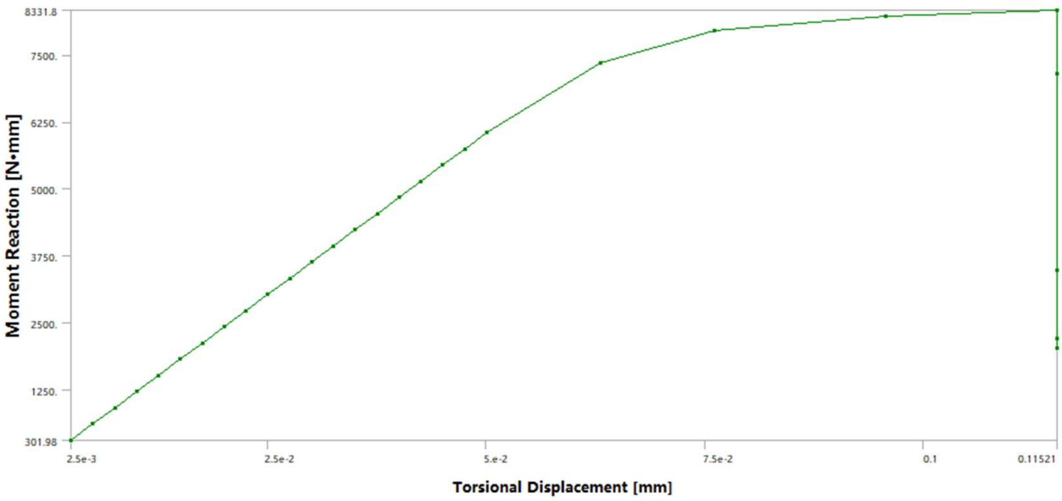

A: Static Structural
Equivalent Plastic Strain
Type: Equivalent Plastic Strain
Unit: mm/mm
Time: 6
2021-03-11 22:47
0.024717 Max
0.021971
0.019224
0.016478
0.013732
0.010985
0.008239
0.0054926
0.0027463
0 Min
ANSYS
2020 R2
ACADEMIC
0.000
4.500
9.000 (mm)
2.250
6.750
Y
X
Z



EQUIVALENT PLASTIC STRAIN RESULTS FROM 3S TO 6S

The above figure shows the variation of the equivalent plastic strain of the specimen during the fracture process. It can be seen that the fracture of the specimen gradually extends from the periphery to the interior, and the fracture is very flush. The specimen is not completely torsionally fractured due to the relatively coarse meshing in this case, and the results would not converge if a larger torsional displacement is applied. Nevertheless, the simulation clearly demonstrates the torsional fracture process of a ductile material and gives laboratory-compatible results.

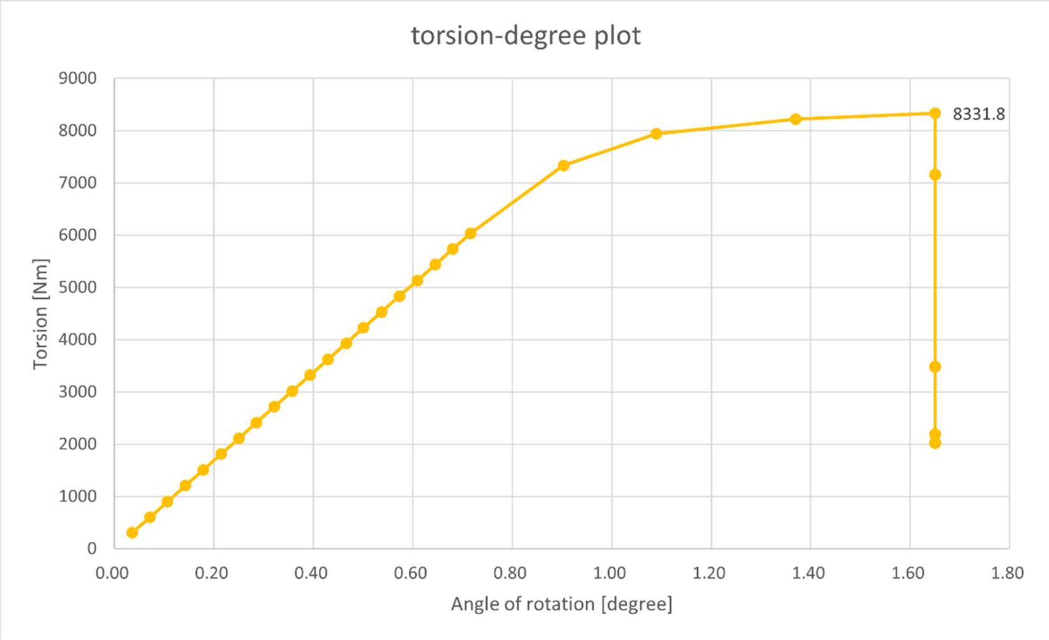
3. Torsion-rotation diagram.



In the laboratory, the computer or the output machinewould output a torsion diagram as shown in FIG 4. However, as described in the previous section on the setting of boundary conditions, the torsion used in this paper is actually a displacement based on a columnar coordinate system applied to the outer surface of the torsion end of the specimen, so the output that can be obtained in ANSYS is shown above, with the horizontal coordinate being the displacement in units of [mm] of displacement. In fact, this plot is already very close to the theoretical laboratory model, but to facilitate comparison with the theoretical model, the data can be exported to excel, using the arc length versus angle

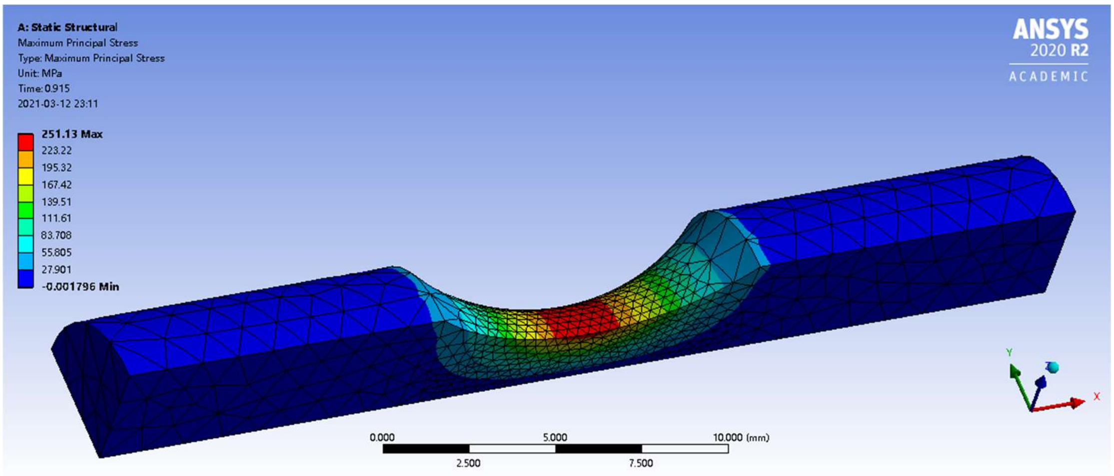
$$L = \alpha \cdot r = n \cdot \pi \cdot \frac{r}{180}$$

The torsion-degree plot is obtained as shown in the following figure.



Brittle material

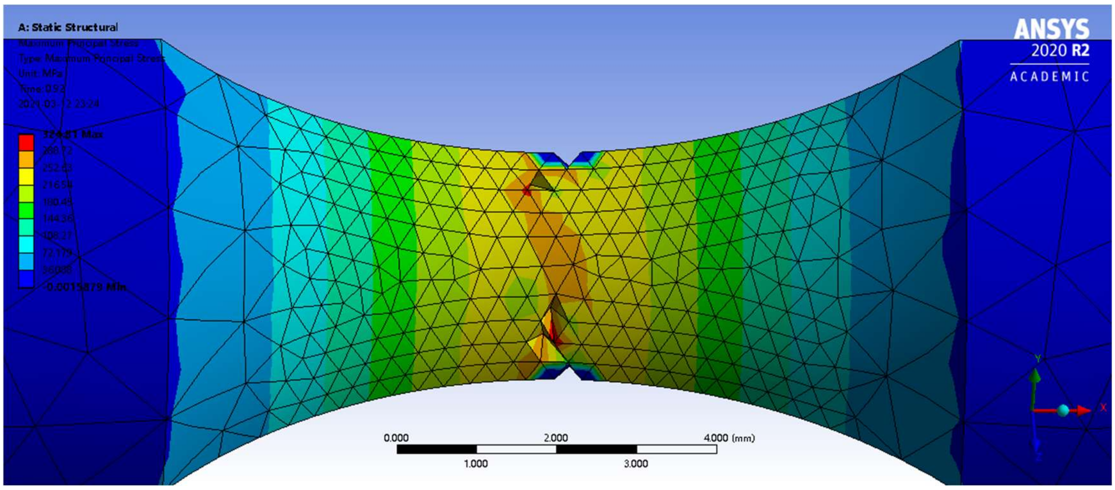
1. Before failure.



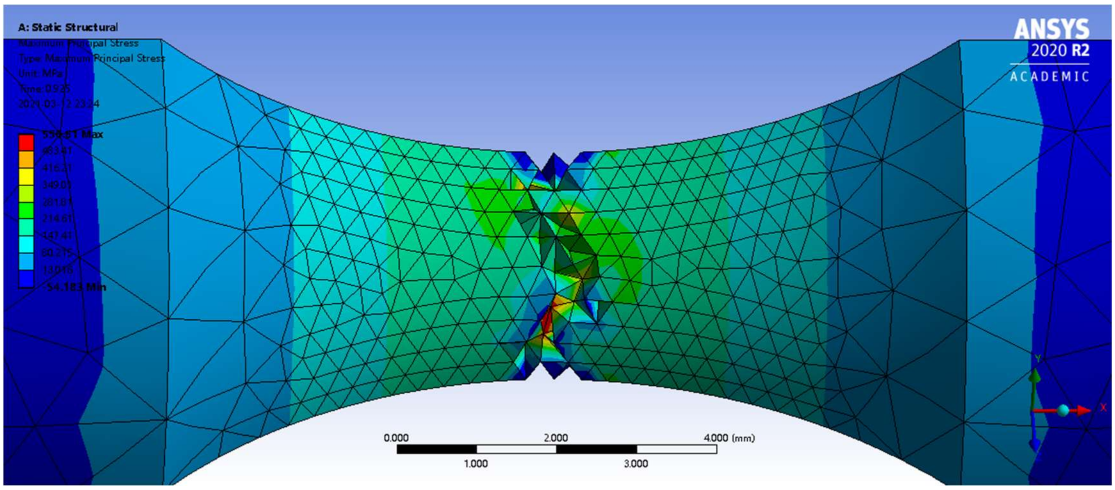
MAXIMUM PRINCIPAL STRESS BEFORE FAILURE

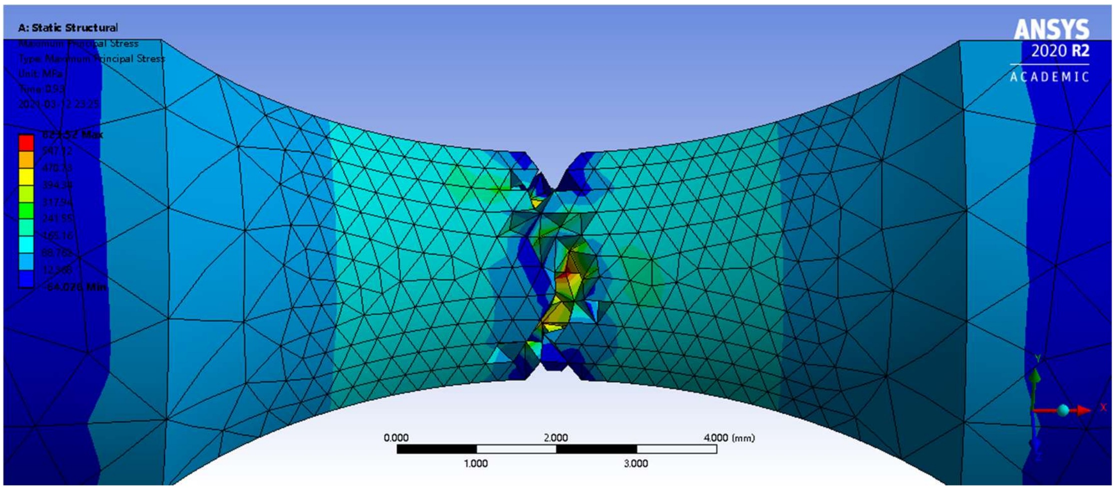
As shown in the figure, torsion of a brittle material still decreases the stress and strain from the periphery to the interior in the same way as torsion of a ductile material.

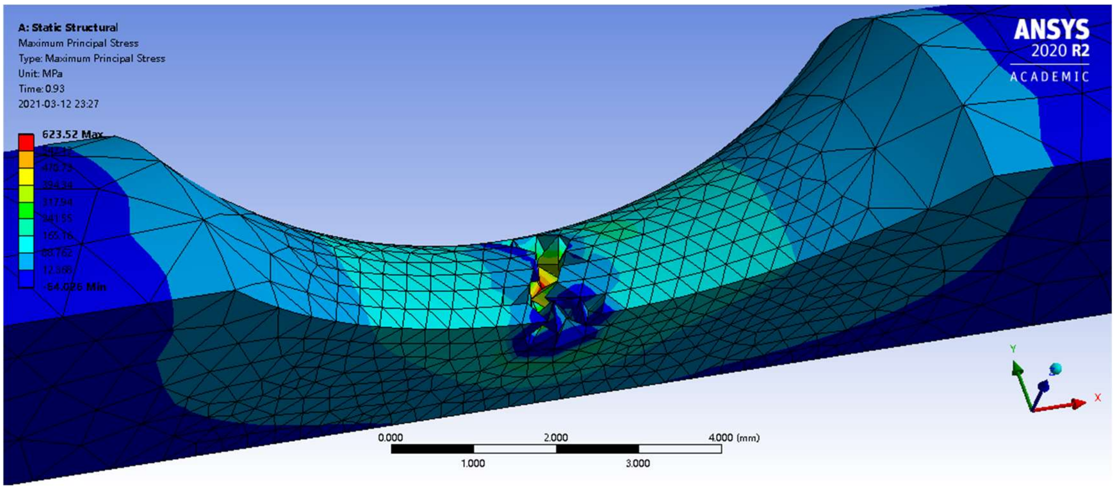
2. Failure process.



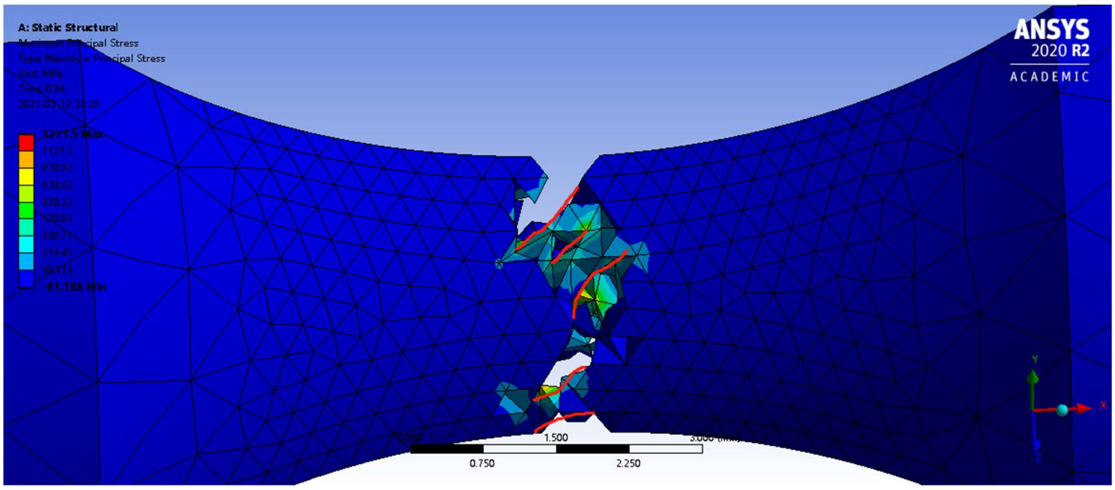
Maximum Principal Stress at 1.02 degrees



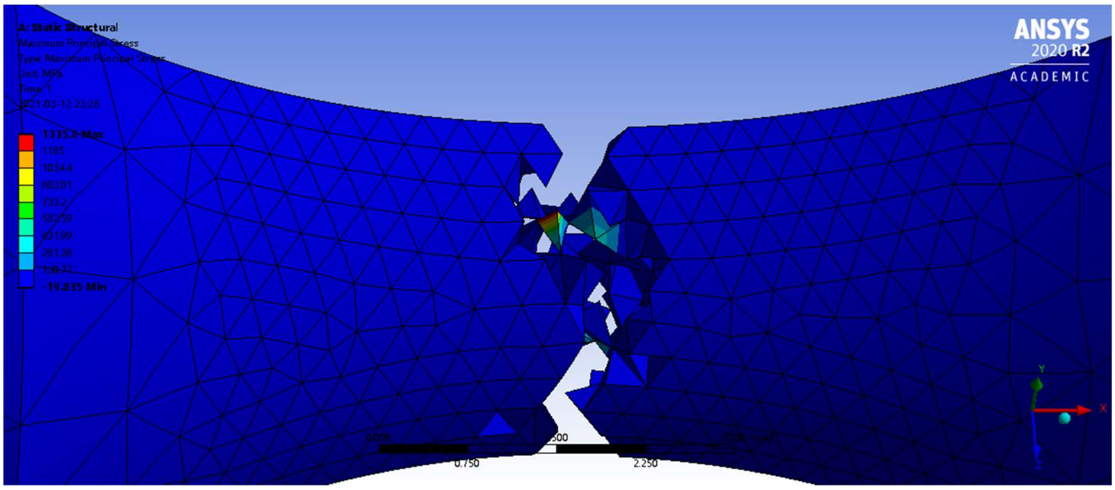




MAXIMUM PRINCIPAL STRESS AT 1.03 DEGREES



MAXIMUM PRINCIPAL STRESS AT 1.05 DEGREES



MAXIMUM PRINCIPAL STRESS AT 1.32 DEGREES

Above is the result of maximum principal stress.

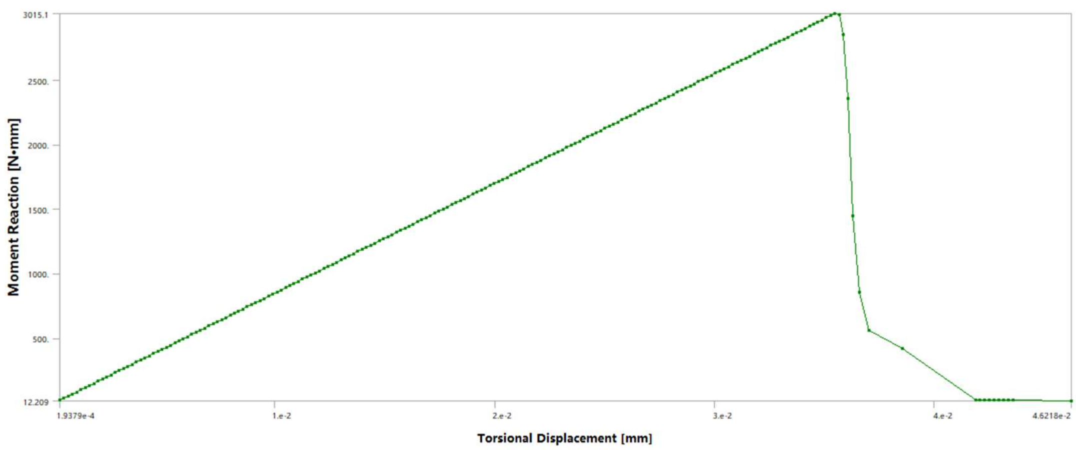
At 1.02 degrees, the specimen starts to fracture. The red and orange parts of the cloud plot show a very obvious slope angle, which indicates that there is a certain slope at this time when the maximum stress is concentrated on the outer surface, which is consistent with the laboratory results. Also, it can be seen that fracture will start from an arbitrary point when the material is free from cell defects.

After 1.02 degrees, the fracture continues. The cloud plot from another angle at 1.03 degrees shows that the fracture of the specimen occurs first from the outer surface and fractures gradually inward.

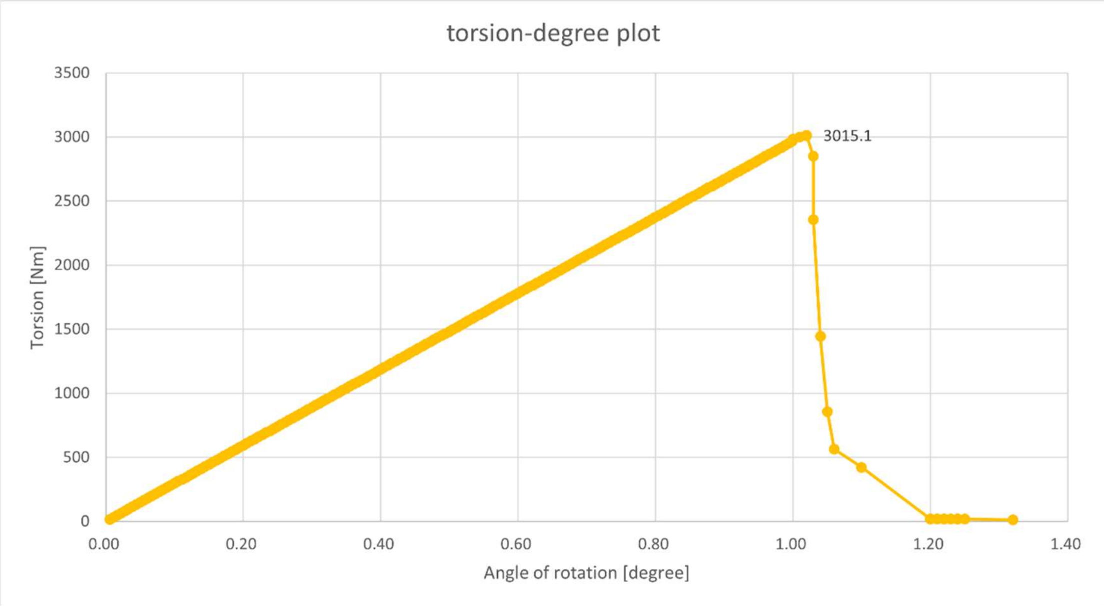
I have marked the elements that are still attached on the 1.05 degrees plot because the color is not too obvious. As you can see, since the fracture occurs randomly, there are still elements attached to the outer surface when the inner part of the specimen has already started to fracture. This is an aspect of torsional fracture of brittle materials that is different from that of ductile materials.

Eventually, the whole specimen is twisted off, as shown in the cloud diagram at 1.32 degrees.

3. Torsion-degree diagram



As with the torsion simulation for ductile materials, the results for brittle materials are still only available as moment-displacement diagrams, which show a linear relationship between the moment and displacement of the specimen before fracture, and a corresponding abrupt decrease in moment when the specimen fractures when torsion reaches its highest point. Putting the resultant data into excel for re-editing, the corresponding moment-degree diagram can be obtained as shown below.



Chapter 3: Conclusion

In this thesis, the theoretical models of tensile test, torsion test and bending test as well as the operating conditions and principles of laboratory measurements are reviewed, and five sets of theoretical bases are obtained before the simulation. Based on these five sets of theoretical bases, the corresponding material failure processes were simulated in the finite element modelling software, i.e. ANSYS.

The five sets of models are the fracture simulation of ductile and brittle materials for tensile test, the fracture simulation of ductile and brittle materials for torsion test, and the plastic deformation simulation when the bending test exceeds the elastic limit. All five sets of simulations yielded results very close to those of laboratory operations.

In the tensile test and torsion test, the difficulty that this paper tries to break through is how to obtain the ideal fracture simulation based on the default implicit algorithm of ANSYS when the simulation does not have sufficient fracture failure criterion. The approach used in this paper is to add APDL command in Analysis Setting, which uses a series of logical statements for cyclic comparison, and APDL EKILL command statements to kill the elements on the model that exceed the preset strength and keep them from showing up in the results.

In order to make this approach work effectively, this paper deals with several related difficulties.

The first difficulty is that the APDL command increases the difficulty of converging the computational results, and therefore requires as many computational steps as possible within the computing power of the computer. The benefit of this is twofold. The first one is that, in general, the increase of computation steps can help to reduce computation errors. On the other hand, always monitoring the convergence state in the solution information can artificially end the computation before it is about to fail to converge, and usually by this time, relatively satisfactory results can already be obtained. If the result is not satisfactory, you can also save computation time and advance to the next attempt. Another way to help convergence is to obtain a more accurate amount of fracture tension or torsion through debugging. This is because, unlike laboratory measurements, when the amount of tension or torsion used as a boundary condition in the software is beyond the fracture point, the software will regard it as an unreasonable condition setting and thus give results that cannot be continued.

The second difficulty is that the software used in this thesis is the free student version of ANSYS, which has a limitation on the calculation volume, i.e., it can only allow models with a maximum of 30,000 nodes to participate in the calculation. Therefore, in order to make the calculation results more accurate, this paper adopts the 1/4 model approach to make the mesh as fine as possible. For the torsion test, the symmetric tool must be used because of the torsional moment, which increases the number of nodes, and therefore adjusting the mesh becomes one of the difficulties.

For the fracture simulation of ductile materials, a geometric model with little variation in cross-sectional area is used in this paper. Since the outer surface of the model is homogeneous, when using uneven meshing will further increase the difficulty of computational convergence, so only homogeneous meshing can be used for ductile materials, and the final results are relatively coarse.

For the fracture simulation of brittle materials, the hourglass shaped specimen with a large variation of cross-sectional area is used in this paper, and since there are large inhomogeneities on the surface of the model, a finer mesh can be arranged at the center of the specimen of interest to obtain more accurate results.

Chapter 4: Future work

As stated in the conclusion, this thesis still has some shortcomings in several aspects.

One is that the limitation of the free student version of ANSYS on the number of meshes results in less accurate simulations for both simulations of ductile materials. If the number of meshes can be increased using the commercial version of the software, it is expected that more beautiful fracture surface simulations and more accurate fracture processes will be obtained.

Another point is that, as mentioned in the previous article on Failure Analysis of brittle materials, when the material cannot be completely fractured and there are still a few elements connected together in a point-to-point manner, the very small stress area will result in a very large stress, which will produce unreasonable data in the resultant cloud. This is also caused by the lack of fine mesh division.

In addition, the subsequent work can try to use multilinear material for the simulation of ductile materials. The use of materials with significant yield plateaus and longer strengthening processes will result in more pronounced necking in tensile tests.

Acknowledgement

Many thanks to my supervisor, Professor Luca Goglio, who always gave me a lot of useful advice. Also, a sincere thank you to my friends Yuhui Li, Xun Song and Linglong Ma, who helped me a lot with my thesis and always calmed me down when I was anxious. Finally, I would like to thank my parents, who have given me great emotional and financial support.

Bibliography

[1] [Online]. Available: http://me1.aut.ac.ir/staff/solidmechanics/alizadeh/Tensile%20Testing.htm.

[2] Budianto, M. T. Wahyudi, U. Dinata, Ruddianto and M. M. E. P., "Strength Analysis on Ship Ladder Using Finite Element Method," *Journal of Physics Conference Series,* 2018.

[3] J. R. Davis, "Mechanical Behavior of Materials under Tensile Loads," in *Tensile Testing, 2nd Edition*, ASM International, 2004, p. 29.

[4] 博弈创作室, APDL 参数化有限元分析技术及其应用实例, 中国水利水电出版社, 2004.

[5] peteroznewman, "I want to SEE the failure!," Ansys Learning Forum, May 2018. [Online]. Available: https://forum.ansys.com/discussion/1373/i-want-to-see-the-failure.

[6] A. HELP, "EKILL," Ansys HELP, [Online]. Available: https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v202/en/ans_cmd/Hlp_C_EKILL.html.

[7] Sih G. C. and Liebowitz H., "On the Griffith energy criterion for brittle fracture," *International Journal of Solids and Structures,* 1967.

[8] Sih G. C. and Macdonald B., "Fracture mechanics applied to engineering problems-strain energy density fracture criterion.," *Engineering Fracture Mechanics,* 1974.

[9] A. I. COURSES, "How to Evaluate Stress and Yielding in Plasticity — Lesson 2," Ansys, [Online]. Available: https://courses.ansys.com/index.php/courses/topics-in-metal-plasticity/lessons/how-to-evaluate-stress-and-yielding-in-plasticity-lesson-2/.