Tutorial 1

BASIC µETA

STATIC ANALYSIS

Table of Contents

1.1. Introduction	2
1.1.1. Prerequisites	2
1.1.2. Problem Description	2
1.1.3. Data Files	2
1.2. Post Processing Procedure	3
1.2.1. Read Model Geometry and Results	3
1.2.2. Handling of entities display	5
1.2.3. Set Undeform states	7
1.2.4. Fringe (contour) plots and options	8
1.2.5. Fringebar (color bar) options	10
1.2.6. Range options	12
1.2.7. Identification of results	14
1.2.8. Statistics	16
1.2.9. Annotations	18
1.2.10. Append Results - Scalar and Vector results simultaneously	21
1.2.11. Report	23
1.2.12. Model Report	26
1.3. Conclusion	27

1.1. Introduction

This tutorial provides an entry overview of basic post-processing steps that could be followed within µETA to conduct a standard Static analysis. Not all µETA capabilities and features are exhausted within this tutorial.

The steps described in this tutorial include: Load the model geometry and results Identify different items of the model Fringe (contour) plots and options Fringebar options Range options Identification of results Statistics Annotations Appending results Scalar and Vector plots simultaneously Report

1.1.1. Prerequisites

It is recommended for users who have never used µETA before to read the Chapter 2 of µETA Users Guide.



1.1.2. Problem Description

The model used for this tutorial is shown in the picture. It consists of 2 plates connected along one of their edge defining an angle of 90° and 2 stripes of bar elements to support the 2 plates. Seven different loadcases (subcases) have been solved. The first three are pure Torsion, Tension and Bending respectively and the last 4 are combinations of the first 3.

1.1.3. Data Files

The files required for this tutorial are

/meta_post_tutorial_files/01-BASIC_STATIC/2_plates_bars.nas
/meta_post_tutorial_files/01-BASIC_STATIC/2_plates_bars.op2

1.2. Post Processing Procedure

1.2.1. Read Model Geometry and Results



📙 🛛 File I	File Manager			
Look in: 🔄 rial_files/01-BAS	IC_STATIC/ 💌 🖙 🗈 (* 🖩 🎹		
Name	Name	Size		
i	2_plates_bars.nas	32916		
	2_plates_bars.op2	577200		
	I	F		
File <u>n</u> ame: 2_plates_bars.nas	;	<u>O</u> pen		
File type: All Supported file	s (*.op2 *.nas * 💌	Cancel		



Start µETA.

From the **Read Results > Geometry** tab, first make sure that the **File Format** option menu is switched to either **Auto detect** or **Nastran**. **1.** linvoke the File Manager. Navigate and select one of the files:

> 2_plates_bars.nas 2_plates_bars.op2

<u>Tips:</u>

The model's geometry can be read either from the input file (NASTRAN Bulk data) or the .op2 file. However loading geometry from the input file (NASTRAN Bulk data) provides extra information for the loads of each subcase.

Press **Open** to read in the model.

The model is read and it should appear as it is shown in the image on the left.

Read Results	×
File Format	Model action
Auto detect	▼ Load to Model:0 ▼
Geometry Resul	ts Curves
Filename	
orials/tutorial_files	s/01-BASIC_STATIC/2_plates_bars.op2 🗾
States Deforma	ation Scalar Vector
SUBCASE 1	::TORSION: SUBCASE 1
SUBCASE 2	::TENSION: SUBCASE 2
SUBCASE 3	::BENDING: SUBCASE 3
SUBCASE 4	::ALL-LOADS COMBINED: SUBCASE 4
SUBCASE 5	::TORSION-TENSION: SUBCASE 5
SUBCASE 6	::TORSION-BENDING: SUBCASE 6
SUBCASE 7	::TENSION-BENDING: SUBCASE 7

Read Results	×
File Format	Model action
Auto detect	Load to Model:0
Geometry Results	Curves
Filename	
orials/tutorial_files/03	1-BASIC_STATIC/2_plates_bars.op2 💌
States Deformatio	n Scalar Vector
Displacements	•
Translational	•

Read Options

 Deformation Scale Factor
 Cut-Off

 Auto calculate
 Small Displacements

Geometry Results Curves	
Filename	
orials/tutorial_files/01-BASIC_STATIC/2_plates_bars.op2	
States Deformation Scalar Vector	
Stresses	•
Stresses * Von Mises	•
Stresses * Von Mises Top and Bottom	•

Default Label	Set as Default
Stresses, Von Mises, Top and Bottom, Corner	See as beladie
+ 🔽 Deformation + 🔽 Scalar + 🗖 Vector	🛷 Read

Switch the **Read Results** card to the **Results** tab. The corresponding *.op2* file is automatically selected and all available subcases are identified in the **States** tab and selected by default.

Tips:

You may deselect from the list the subcases that you do not want to load. For this example, load all subcases.

Switch to the **Deformation** tab. By default, the **Displacements**, **Translational** components are selected for loading.

Tips:

In case other nodal vector data (such as SPC forces, velocities, etc) should be loaded as Deformation results, this can be set here through the 2 option menus.

Select from the **Read Options > Deformation Scale Factor** menu, at the bottom of the **Deformation** tab, to the **Auto calculate** option.

<u>Tips:</u>

Using this option, μ ETA will calculate and assign, upon loading, an appropriate Scale Factor for the visualization of deformations (which is necessary in this case since this is a static analysis). In such cases, it is recommended to use the **Auto** calculate.

Switch to the **Scalar** tab. 4 menus appear. The first one holds all available types of results (stresses, strains, etc). By default, this is switched to **Stresses**.

The second menu includes all available types / components for the results type that is selected in the first menu. Switch this second menu to **Von Mises** option.

The third menu includes all available options for the location of results related to section thickness direction of shell elements. Switch this menu to the **Top and Bottom** option to load results on both sides of shells.

The fourth menu includes all available options regarding the location of results within the element (Centroid, Integration Points, Corner). Switch this menu to **Corner** option.

Press **Read**, at the bottom of the **Read Results** card, to load the selected **Deformation** and **Scalar** results for the selected subcases.

1.2.2. Handling of entities display.

	×
	-
M SOLIDS	
CBEAM	CROD
CTUBE	PLOTEL
🕅 RBE3	M RBAR
CONM	CBEND
CDAMP	M CMASS
CVISC	CGAP/CWELD
PLANES	₩ ISO
FREE NODES	
SPC1	MPC
MOMENT	PLOAD
None	Invert
	SOLIDS CEBAM CTUBE CTUBE COM COMM COAMP COAMP COSSC PLANES FREE NODES FREE NODES FSCL MOMENT None

Press F12 key to open the Set Visible Entities card. Press All button so as to enable the visualisation of boundary conditions which are not by default visible.

The green lines are the Forces and it is obvious that a different scale factor has to be set for the better visualisation of them.





- 1. Activate the Wire drawing style.
- 2. Press the Set Transparency button.
- 3. Select the horizontal plate.

The PLOADs (yellow arrows) can be clearly seen. These PLOADs are related to the Bending loadcase.

Right click on the **Set** Transparency button and select the same Pid to become non-transparent again.





New Identify Settings Identify Settings Image: Settings Image: Settings Image: Setings Image: Settings	Part Manager						×
Image Visible Color Module Id Version 0./motival 0./motival 0 0./motival 0./motival 0 0./motival 0./motival 0 0./motival 0 0 0./motival 0	New Identify	Settings					
Name Visible Color Module Id Version 0 · Orimntrial diskrittanas/meta_GENERAL/DOCS_META/ * · O · SUBCASE_1 · O · O · · · · · · · · · · · · ·	€ ⇒ ↑ C					Ē	- 1
Ormutraid disk/titanas/meta_GENERAL/DOCS_META/ O COMMON COMMON Common and a second a	Name		Visible	Color	Module Id	Vers 🖓 🗸	• ↓
COMMON O Model Tutorial 1 O SUBCASE 1 O SUBCASE 3 O SUBCASE 3 O SUBCASE 5 O SUBCASE 6 O	≜- 0:/mnt/raid_	disk/titanas/meta_GENERAL/DOCS_META/	<u> </u>				_
Model_Tutorial_1 0 +	COMMON	1	<u> </u>				0
	Model_Tu	itorial_1	<u> </u>				0
Image: subcase_3 0 Image: subcase_4 0 Image: subcase_5 0 Image: subcase_6 0 <t< td=""><td>- SUBCASE</td><td>_1</td><td><u> 🔆</u></td><td></td><td></td><td></td><td>0</td></t<>	- SUBCASE	_1	<u> 🔆</u>				0
	SUBCASE	_2	<u>ې</u>				0
	- SUBCASE	_3))				0
	- SUBCASE	_4	<u>)</u>				0
	- SUBCASE	_5	<u>)</u>				0
Itotal 22 selected 2 Name Value Image: Additional selected and the	I SUBCASE	_6	۰.				0
Image: Selected 2 Name Value Module Id Name Model_Tutorial_1 Version Image: Selected 2 Study Version 0							
Item Item <th< td=""><td></td><td></td><td></td><td></td><td></td><td></td><td></td></th<>							
Name Value Module Id Model_Tutorial_1 Version Representation Study Version 0	[•]			lanta	1 22	a da ata di D	
Name Value Module Id	-			tota	al 22	selected 2	_
Module Id Model_Tutorial_1 Version Representation Study Version 0	Name	Value		_			
Name Model_Tutorial_1 Version	Module Id						
Version Representation Study Version 0 v	Name	Model_Tutorial_1					
Representation Study Version 0	Version						
Study Version 0	Representation						
	Study Version	0		-			

From the **Tools** pull-down menu open the **Settings** window and set a new **Load Scale** factor within the **Windows Settings>General** tab (the default is 1). Here a value 0.1 is applied.

All loads are now scaled and the Forces are clearly displayed. These Forces are related to the Torsion and the Tension loadcases.

Tips:

The length of the arrows representing the loads, is relative to the load value.

Provided that you have loaded geometry from the input file, the loads and the boundary conditions are grouped in μ ETA according to the subcase they correspond to.

Open the Part Manager



Select for example the **SUBCASE_2** and **Model_Tutorial_1** groups from the list with Right Mouse Button and from the menu that appears select the **Or** option to view the model with the Tension forces.



1.2.3. Set Undeform states





The model with the Tension forces.

1. Open **Tools>States** and in the **Undeform** tab of the **States** list press the **Add** button so as to set the ORIGINAL STATE as the **Undeform** state that will be displayed always.

2. Apply a uniform **User color** to the **Undeform** state. To do this, activate the **User color** toggle button.

3. Select the **Palette** button to apply a color other than the default from the color selector (here the red color is applied).

Tips:

You can create more than one **Undeform** states by selecting the state you want to be displayed as undeform and pressing the **Add** button again. **Undeform** states will be displayed in the relevant list.

4. Select SUBCASE 4 (this subcase is the combination of the first 3 subcases) in the **States** list.

<u>Tips:</u>

You can change states using:

- The Animation scroll-bar (5).
- The **Up** and **Down** keys when the focus is in the **States List**

- The **Page Up** and **Page Down** keys when the focus is on the model

- **Home** and **End** keys navigate the model to the ORIGINAL and the last state respectively.

以上内容仅为本文档的试下载部分,为可阅读页数的一半内容。如 要下载或阅读全文,请访问: <u>https://d.book118.com/02532314410</u> <u>3011131</u>