Tutorial 1

BASIC μETA

STATIC ANALYSIS

Table of Contents

1.1. Introduction

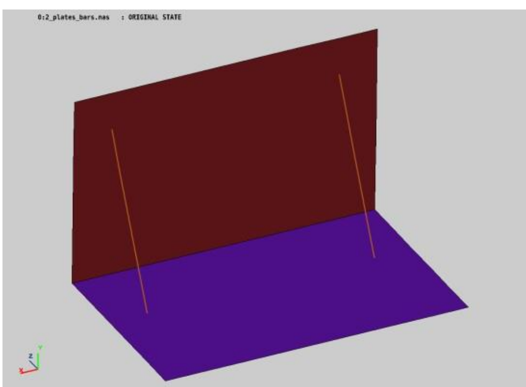
This tutorial provides an entry overview of basic post-processing steps that could be followed within μETA to conduct a standard Static analysis.
Not all μETA capabilities and features are exhausted within this tutorial.

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



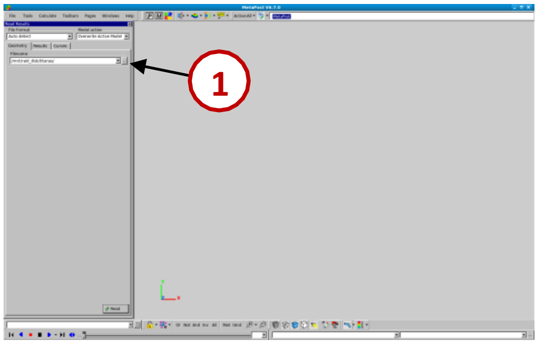
The model used for this tutorial is shown in the picture. It consists of 2 plates connected along one of their edge defining an angle of 90° and 2 stripes of bar elements to support the 2 plates. Seven different loadcases (subcases) have been solved. The first three are pure Torsion, Tension and Bending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

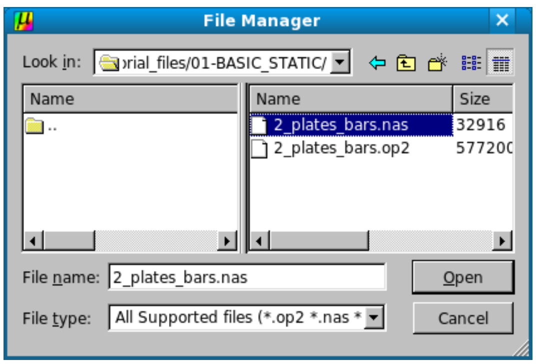
1.2. Post Processing Procedure

1.2.1. Read Model Geometry and Results



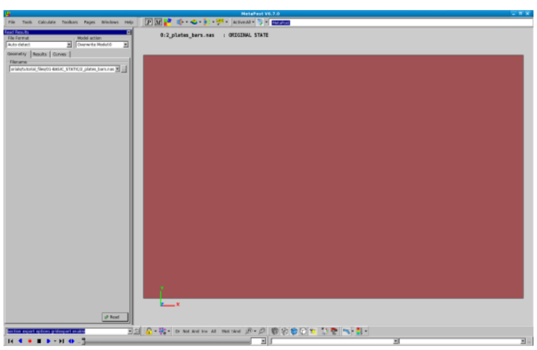
Start μETA.
From the **Read Results > Geometry** tab, first make sure that the **File Format** option menu is switched to either **Auto detect** or **Nastran**.
1. Iinvoke the File Manager. Navigate and select one of the files:

```
2_plates_bars.nas
2_plates_bars.op2
```

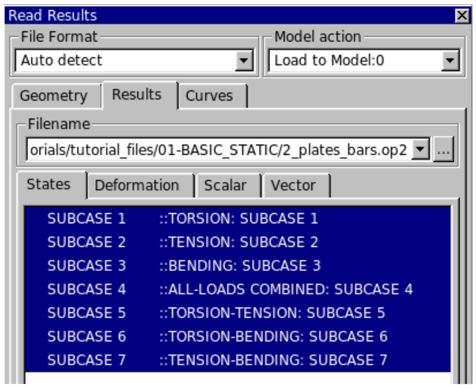


Tips:
The model's geometry can be read either from the input file (NASTRAN Bulk data) or the .op2 file. However loading geometry from the input file (NASTRAN Bulk data) provides extra information for the loads of each subcase.

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

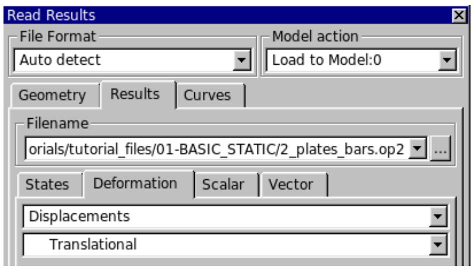


The model is read and it should appear as it is shown in the image on the left.



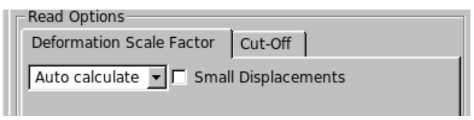
Switch the **Read Results** card to the **Results** tab. The corresponding *.op2* file is automatically selected and all available subcases are identified in the **States** tab and selected by default.

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



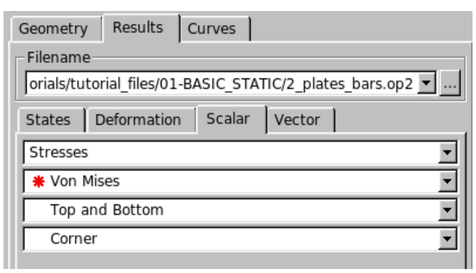
Switch to the **Deformation** tab. By default, the **Displacements**, **Translational** components are selected for loading.

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.

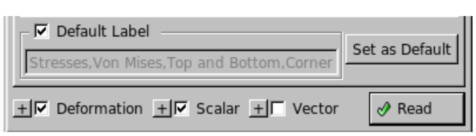


Select from the **Read Options > Deformation Scale Factor** menu, at the bottom of the **Deformation** tab, to the **Auto calculate** option.

Tips:
Using this option, μETA 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.**

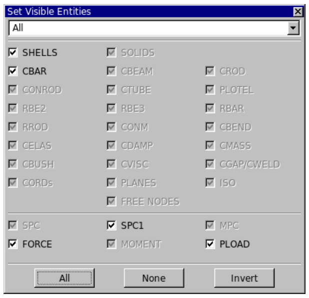


Switch to the **Scalar** tab. 4 menus appear.
The first one holds all available types of results (stresses, strains, etc). By default, this is switched to **Stresses**.
The second menu includes all available types / components for the results type that is selected in the first menu. Switch this second 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.

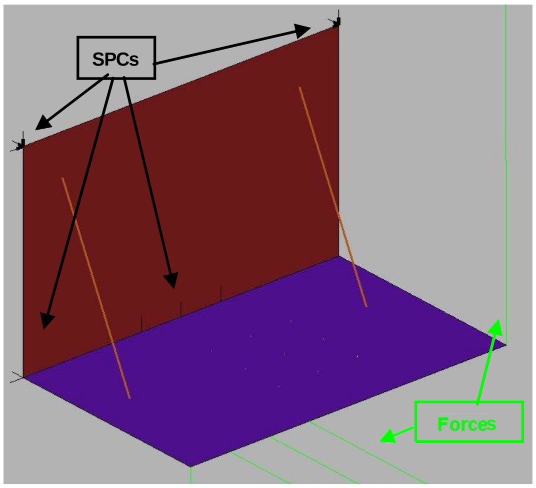


Press **Read**, at the bottom of the **Read Results** card, to load the selected **Deformation** and **Scalar** results for the selected subcases.

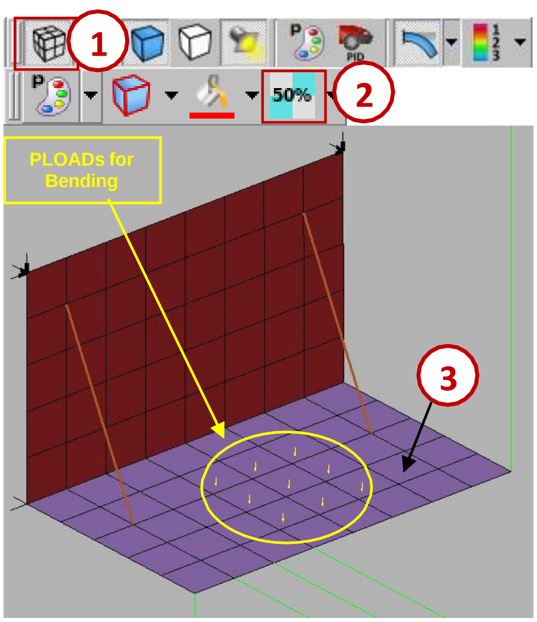
1.2.2. Handling of entities display.



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.



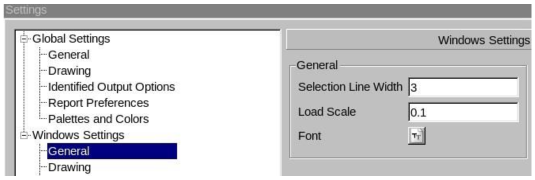
1. Activate the **Wire** drawing style.

2. Press the **Set Transparency** button.

3. Select the horizontal plate.

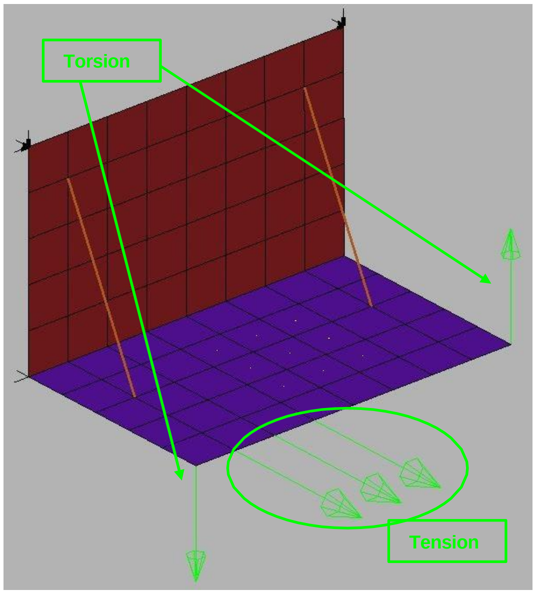
The PLOADs (yellow arrows) can be clearly seen. These PLOADs are related to the Bending loadcase.

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

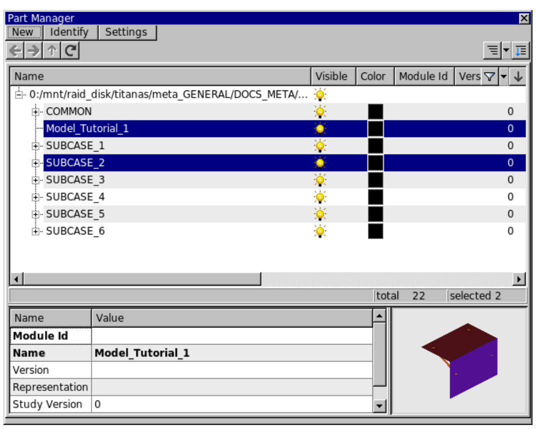


From the **Tools** pull-down menu open the **Settings** window and set a new **Load Scale** factor within the **Windows Settings>General** tab (the default is 1). Here a value 0.1 is applied.

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

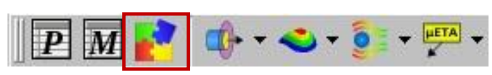


Tips:
The length of the arrows representing the loads, is relative to the load value.

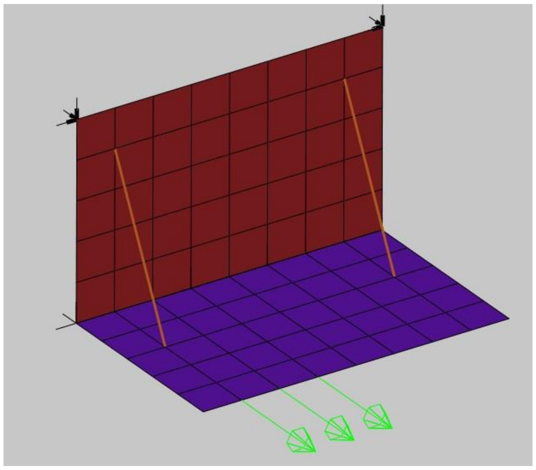


Provided that you have loaded geometry from the input file, the loads and the boundary conditions are grouped in μETA according to the subcase they correspond to.

Open the **Part Manager**

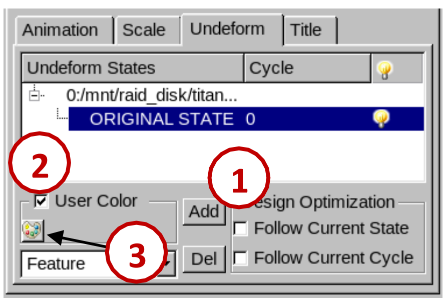


Select for example the **SUBCASE_2** and **Model_Tutorial_1** groups from the list with Right Mouse Button and from the menu that appears select the **Or** option to view the model with the Tension forces.



The model with the Tension forces.

1.2.3. Set Undeform states

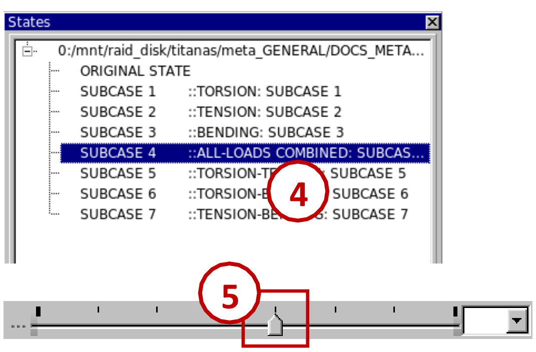


1. Open **Tools>States** and in the **Undeform** tab of the **States** list press the **Add** button so as to set the ORIGINAL STATE as the **Undeform** state that will be displayed always.

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.

Tips:
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.