



UNIMORE
UNIVERSITÀ DEGLI STUDI DI
MODENA E REGGIO EMILIA



Dipartimento di Ingegneria
“Enzo Ferrari”

Progettazione Assistita di Organi di Macchine

Sara Mantovani
sara.mantovani@unimore.it

Agenda

Finite Element Modelling

MSC Marc Mentat

Thin-walled profile in torsion

Thin-walled profile in torsion UPDATE

Mesh convergence

References

Agenda

Finite Element Modelling

MSC Marc Mentat

Thin-walled profile in torsion

Thin-walled profile in torsion UPDATE

Mesh convergence

References

Finite Element Modelling

What? Why?

The Finite Element Method may be described as a systematic procedure through which any function is approximated by a discrete model.

The function is defined over some region in space over which it is continuous. This discrete model consists of a set of values of the given function at a finite number of points in its domain, together with piecewise approximations of the function over a finite number of subdomains.

These subdomains are known as ***finite elements***, and the approximation to the function is made locally over each finite element.

These local approximations are uniquely defined in terms of the discrete values of the function at preselected points on the elements known as ***nodal points***, or simply ***nodes***.

Finite Element Modelling

What? Why?

The finite element model is constructed using the following steps:

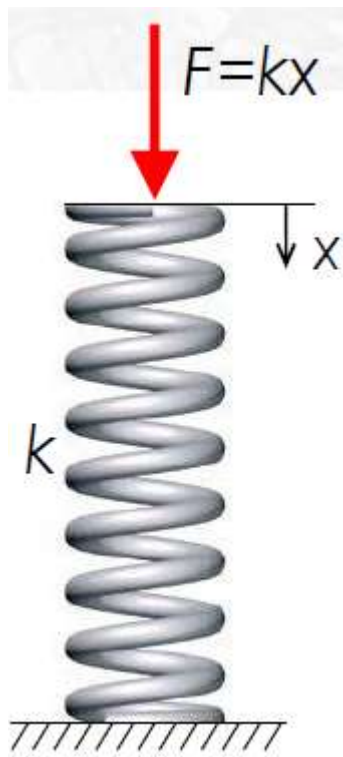
- A finite number of points are identified in the function's domain, and the values of the function specified at these points. These points are generally known as **nodal points**, or simply **node**.
- The domain is divided into a finite number of subdomains called **finite elements**. These are usually of a simple shape, for example triangles or rectangles in two-dimensional space, and tetrahedra or hexahedral in three dimensions. The model of the actual domain can be considered as an assemblage of the finite elements connected together appropriately at the nodes on their boundaries.
- The given function is approximated locally over each element by continuous functions which are uniquely defined in terms of the values of the function (and possibly its derivatives) at the nodes on each element.

Finite Element Modelling

What? Why?

Finite Element formulation defines a relationship between **external forces** applied to certain points of a generic element, and the **displacements** that these forces induce at the same points.

The points that are used to apply forces and evaluate the corresponding displacements are called **nodes**.



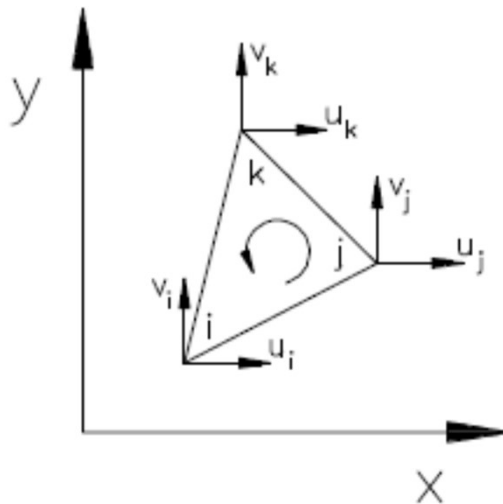
$$F=kx$$

Actually, a single finite element could be compared to a simple **spring** where the force is equal to the stiffness times the displacement.

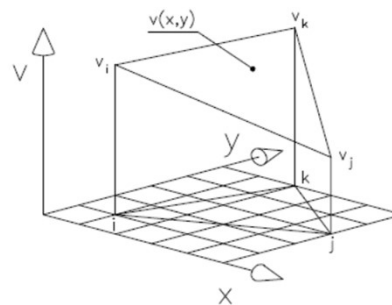
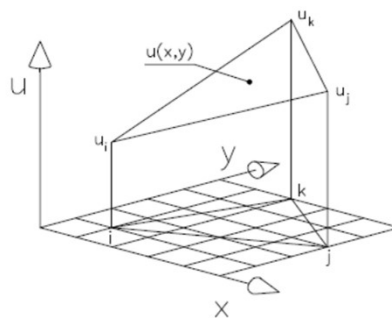
The “only” difference is related to the fact that a simple spring usually exhibits one degree of freedom while a generic finite element exhibits a higher number of degrees of freedom (depending on the element type).

Finite Element Modelling

What? Why?



In the following, the stiffness matrix of the simple plane triangular element is derived. A typical triangular element is shown in figure with corner nodes i , j and k numbered in counterclockwise order. The (\mathbf{x}, \mathbf{y}) displacement components at nodes i , j and k are $(\mathbf{u}_i, \mathbf{v}_i)$, $(\mathbf{u}_j, \mathbf{v}_j)$ and $(\mathbf{u}_k, \mathbf{v}_k)$ respectively.



$$u(x, y) = \alpha_1 + \alpha_2 x + \alpha_3 y$$

$$v(x, y) = \alpha_4 + \alpha_5 x + \alpha_6 y$$

Agenda

Finite Element Modelling

MSC Marc Mentat

- Open the software in Linux
- Files
- Open input file
- Save and Save as ...
- Open output file
- Main Menu
- Toolbar
- Toolbar: UTILS
- Mouse gesture

Thin-walled profile in torsion: model 1

Thin-walled profile in torsion: model 1 updated

Mesh convergence

References

MSC Marc Mentat

Open the software in Linux

Open a terminal

- Type *mentat2013.1 -ogl -glflush*
- Press Enter

Marc: pre-processing and post-processing software

Mentat: no-linear solver

MSC Marc Mentat

Files

Input file: **.mud** or **.mfd**

Graphic view

Input file: **.dat**

Text format

Output file: **.t16** or **.t19**

Graphic view

Output file: **.log** or **.out**

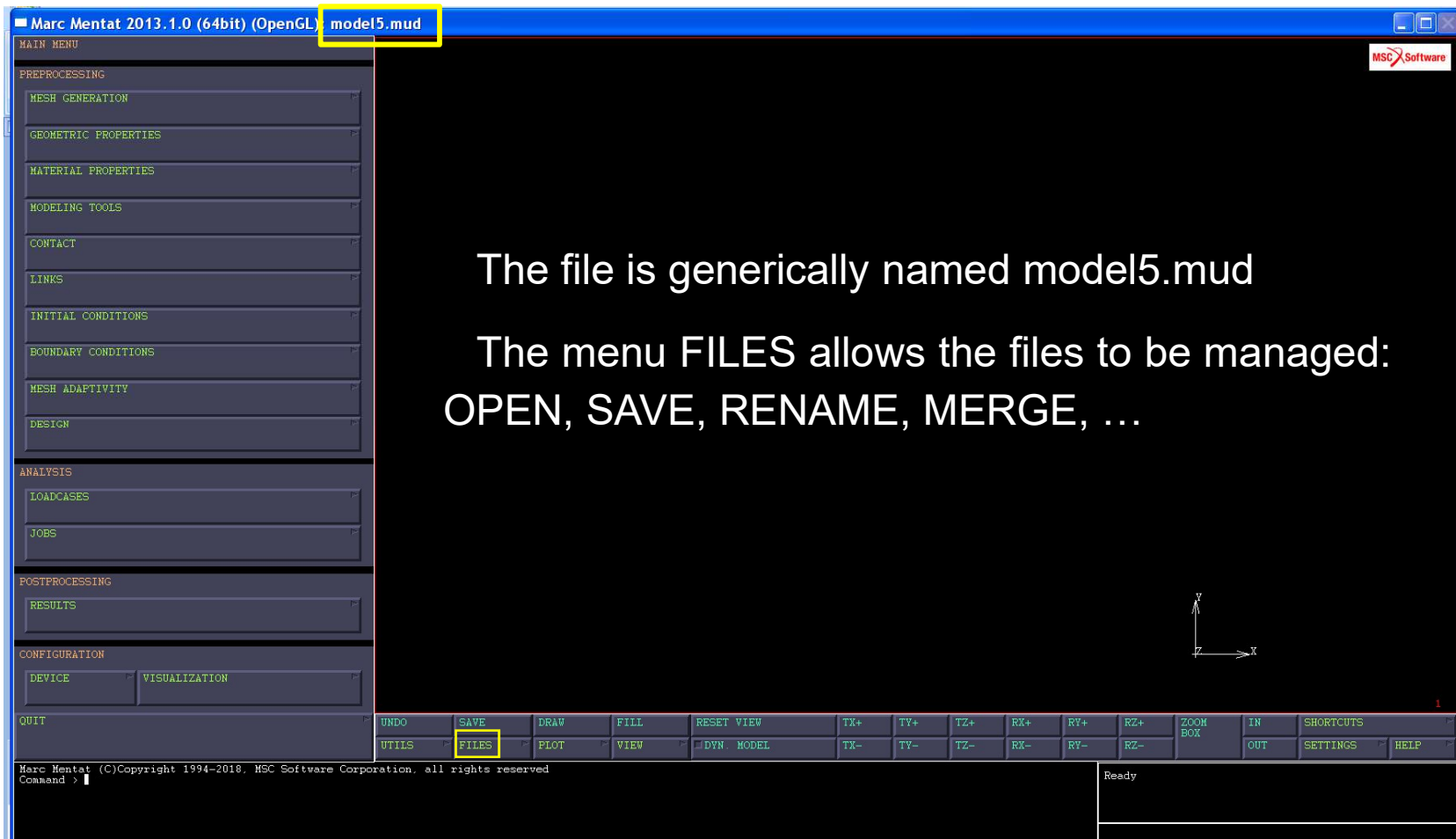
Text format

Optional file: **.proc**

It is a text file that allows the automatic setup model to be built

MSC Marc Mentat

Open Input File



MSC Marc Mentat

Open Input File

It is noted that to manage a file the directory must be specified by the menu **CURRENT DIRECTORY**.

The file path must contain lower case letters and the spacing must be substituted by the _ «underscore».

MSC Marc Mentat

Open Input File

FILE I/O

MODEL

acdel5

TEXT DESCRIPTION

NEW

OPEN

MERGE MERGE RENUMBER

SAVE

SAVE AS DEFAULT STYLE

RESTORE

INTERFACES

IMPORT EXPORT

MARC INPUT FILE

READ OPTIONS

RENUMBER ALL

WRITE

CURRENT DIRECTORY

C:\Documents and Settings\Administrator

EDIT FILE RESET PROGRAM

RETURN MAIN

UNDO SAVE DRAW FILL RESET VIEW TX+ TY+ TZ+ RX+ RY+ RZ+ ZOOM BOX IN SHORTCUTS

UTILS FILES PLOT VIEW /DYN MODEL TX- TY- TZ- RX- RY- RZ- OUT SETTINGS HELP

Command > *open_model "C:\Documents and Settings\Administrator\telaio_v1.mfd"
File is Mentat, version unknown.
Model file C:\Documents and Settings\Administrator\telaio_v1.mfd opened for reading.
Model C:\Documents and Settings\Administrator\telaio_v1 opened.
Command > *new_model
OK to delete current model? yes
Command > |

Ready

MSC Software

e.g.: telaietto_sistema_riferimento.mfd

Marc Mentat Select Model File

OPEN FILE

FILTER *.aud *.mfd

DIRECTORIES

FILES

telaietto_sistema_riferimento.mfd

SELECTION C:\Documents and Settings\Administrator\telaietto_sistema_riferimento.mfd

CANCEL RESET RESCAN OK

MSC Marc Mentat

Save and Save as ...

The screenshot displays the MSC Marc Mentat 2013.1.0 (64bit) (OpenGL) interface. The main window title is 'model5.mud'. The 'FILE I/O' menu is open, with 'SAVE' and 'SAVE AS' options highlighted by yellow boxes. A 'Marc Mentat Select Model File' dialog box is overlaid on the main window, showing a file list with the following entries:

DIRECTORIES	FILES
...	model1.mud
altair	model2.mud
gimp-2.8	model3.mud
idlerc	model4.mud
imagej	pippo.mud
ipython	telaio_v1.mud
matplotlib	telaio_v1.mfd
pdfsam	
thumbnails	
unison	

The 'SELECTION' field in the dialog box contains the path: `C:\Documents and Settings\Administrator\telaio_v1.mfd`. The dialog box has buttons for 'CANCEL', 'RESET', 'RESCAN', and 'OK'. The main window's command line at the bottom shows the following commands and output:

```
Command > *open_model "C:\Documents and Settings\Administrator\telaio_v1.mfd"  
File is Mentat, version unknown.  
Model file C:\Documents and Settings\Administrator\telaio_v1.mfd opened for reading.  
Model C:\Documents and Settings\Administrator\telaio_v1 opened.  
Command > *new_model  
OK to delete current model ? yes  
Command > |
```

MSC Marc Mentat

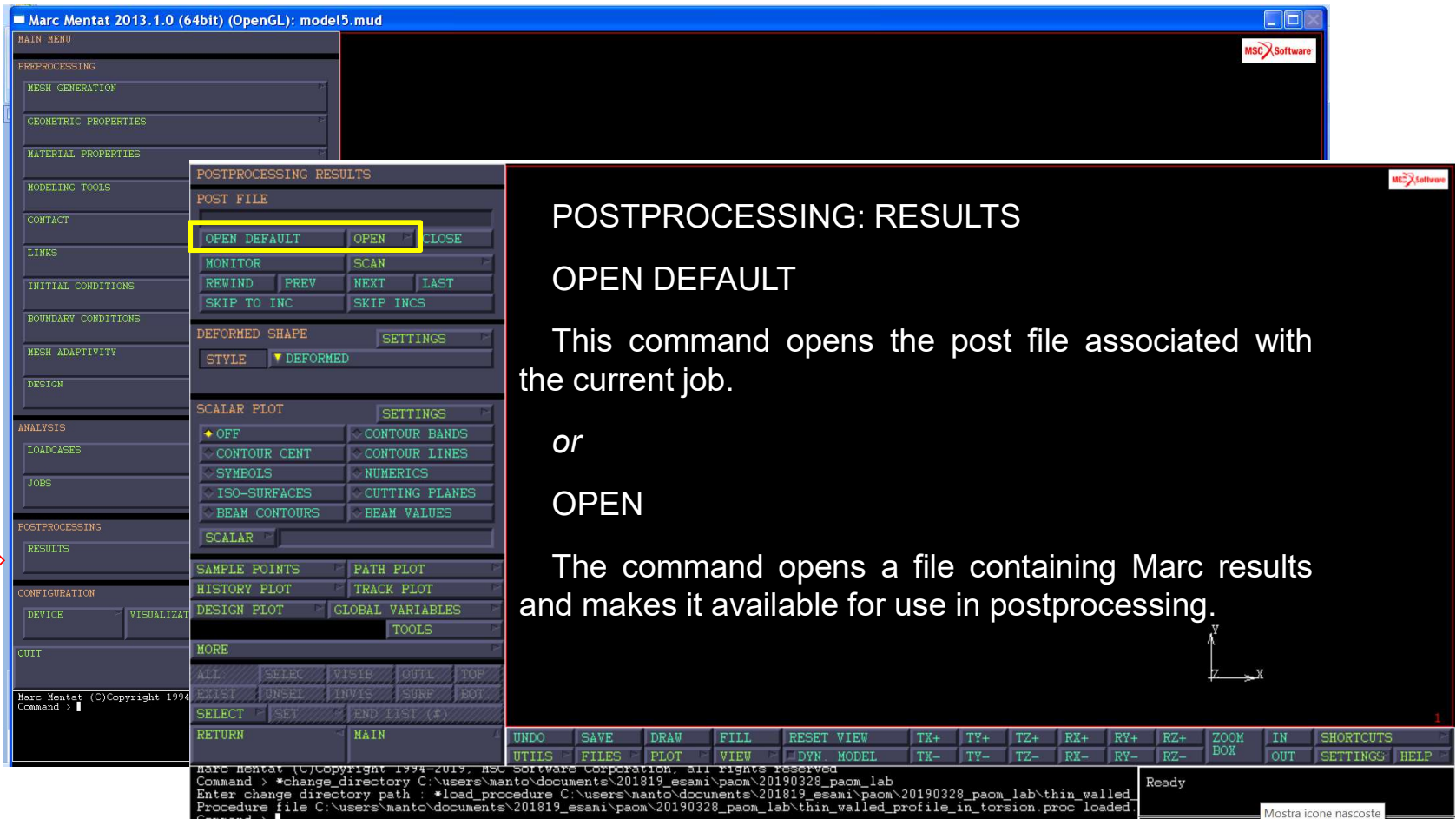
Save and Save as ...

The screenshot displays the MSC Marc Mentat 2013.1.0 (64bit) software interface. The main window title is "Marc Mentat 2013.1.0 (64bit) (OpenGL) : model5.mud". The "MODEL" menu is open, showing options like "NEW", "OPEN", "MERGE", "SAVE", "SAVE AS", and "RESTORE". The "SAVE" and "SAVE AS" options are highlighted with a yellow box. A "Marc Mentat Select Model File" dialog box is open, showing a list of files in the "FILES" section, including "model1.mud", "model2.mud", "model3.mud", "model4.mud", "pippo.mud", "telaletto_sistema_riferimento.mfd", and "telaio_v1.mfd". The "SELECTION" field shows the path "C:\Documents and Settings\Administrator\telaletto_sistema_riferimento.mfd". The "OK" button is highlighted with a yellow box. The status bar at the bottom shows "Ready".

Save is used to overwrite an existing file;
Save as ... is used to save a new file or a file with a new name;
in a well defined directory.

MSC Marc Mentat

Open Output File



POSTPROCESSING: RESULTS

OPEN DEFAULT

This command opens the post file associated with the current job.

or

OPEN

The command opens a file containing Marc results and makes it available for use in postprocessing.

Command > *change_directory C:\users\manto\documents\201819_esami\paom\20190328_paom_lab
Enter change directory path : *load_procedure C:\users\manto\documents\201819_esami\paom\20190328_paom_lab\thin_walled
Procedure file C:\users\manto\documents\201819_esami\paom\20190328_paom_lab\thin_walled_profile_in_torsion.proc loaded
Command >

Agenda

Finite Element Modelling

MSC Marc Mentat

- Open the software in Linux
- Files
- Open input file
- Save and Save as ...
- Open output file
- **Main Menu**
- **Toolbar**
- **Toolbar: UTILS**
- **Mouse gesture**

Thin-walled profile in torsion

Thin-walled profile in torsion UPDATE

Mesh convergence

References

MSC Marc

MAIN MENU

The screenshot displays the MSC Marc software interface. The main menu is located on the left side, containing categories such as PREPROCESSING, ANALYSIS, POSTPROCESSING, and CONFIGURATION. The toolbar is located at the bottom of the interface, featuring various icons for file operations, modeling, and analysis. The command prompt is located at the bottom left of the interface.

- MAIN MENU (left hand side, area in red)
- TOOLBAR (Area underlined in green)
- COMMAND PROMPT (Lowest area, highlighted in yellow)

MSC Marc

MAIN MENU

The MAIN MENU consists in four sections:

PRE PROCESSING: create the problem model

ANALYSIS: perform the numerical analysis

POST PROCESSING: exhibit the results

CONFIGURATION: options e.g. view, colours, ...

The order of the main submenus is not random, they follows the rational steps used to setting up a Finite Element Analysis.



MSC Marc

TOOLBAR

UNDO: Torna indietro di una ed una sola digitazione cancellando i suoi effetti.

SAVE: mi consente di salvare i modelli ed i risultati ottenuti delle cartelle che desidero

FILES: permette di aprire e chiudere file .mud

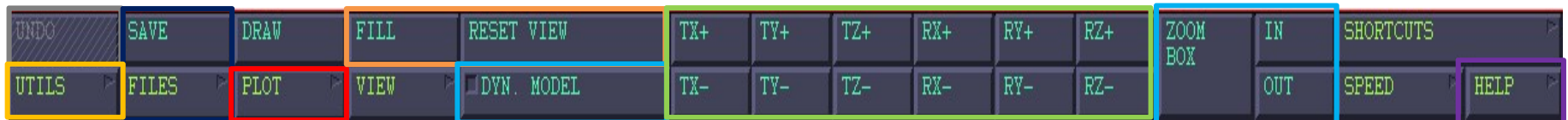
FILL: centra il modello della finestra grafica

RESET VIEW: riporta il modello alla configurazione di partenza sul pino xy

TX+, TX-,..., RZ-: Funzioni che consentono di traslare, ruotare il modello lungo le direzioni del sistema globale X, Y, Z a passi discreti.

ZOOM BOX: mi consente di visualizzare in dettaglio la zona di un modello, di un grafico, ... che ho selezionato

IN e OUT: mi focalizzo o mi allontano dal plot per passi discreti



UTILS: Consente di esportare i dati di output in excel, di misurare distanze (DISTANCE), In questo menu posso selezionare lo sfondo con cui visualizzare la finestra grafica da SNAPSHOT, PREDEFINED CONTOUR MAP,..

PLOT: Consente di plottare o non plottare i nodi, gli elementi, ... e le etichette relative sul modello entrando nei vari sottomenu mediante SETTINGS.

DYNAMIC MODEL: Se attivo consente di spostare il modello nella finestra grafica in particolare:

- ZOOM IN o OUT: tasto dx mouse e si trascina il cursore rispettivamente verso l'interno o l'esterno del modello;

- ROTAZIONE: cliccando il tasto centrale del mouse;
- TRASLAZIONE: premendo il tasto sinistro del mouse.

HELP: raccoglie una vasta documentazione relativa alle caratteristiche del software. Per migliorarne la consultabilità alcuni macro-argomenti sono stati raccolti in Volumi

MSC Marc

TOOLBAR: UTILS

The screenshot displays the MSC Marc software interface. On the left is the UTILITIES menu with various options like POSTSCRIPT, SNAPSHOT, and PROCEDURES. The PROCEDURES option is highlighted with a yellow box. In the center, the 'Marc Mentat Procedure Control' window is open, showing options like CREATE, APPEND, CLOSE, LOAD, EXECUTE, and MENU EXECUTE. The UTILS button in the bottom toolbar is also highlighted with a yellow box. On the right, a 'Marc Mentat Select Procedure File' dialog box is open, showing a list of files including 'thin_walled_profile_in_torsion.proc'. A text overlay on the right side of the image reads: 'UTILS PROCEDURE LOAD thin_walled_profile_in_torsion.proc OK'. The bottom of the screen shows a command prompt with the following text: 'Marc Mentat (C)Copyright 1994-2019, MSC Software Corporation, all rights reserved. Command > *change_directory C:\users\manto\documents\201819_esami\paom\20190328_paom_lab Enter change directory path :'

MSC Marc

TOOLBAR: UTILS

The screenshot displays the MSC Marc software interface. On the left, the UTILITIES menu is visible, with the PROCEDURES option highlighted in yellow. In the center, the Marc Mentat Procedure Control dialog box is open, showing various options for procedure execution. On the right, text explains the functions of the START, STOP, STEP, and QUIT buttons. At the bottom, the software's command line and toolbar are visible, with the UTILS button highlighted in yellow.

START and STOP
run automatically the procedure

STEP
runs the procedure line by line

QUIT
Abandons the procedure execution

Command > *change_directory C:\users\manto\documents\201819_esami\paom\20190328_paom_lab
Enter change directory path : *load_procedure C:\users\manto\documents\201819_esami\paom\20190328_paom_lab\thin_walled
Procedure file C:\users\manto\documents\201819_esami\paom\20190328_paom_lab\thin_walled_profile_in_torsion_proc loaded.
Command >

MSC Marc

TOOLBAR: UTILS

The screenshot displays the MSC Marc software interface. On the left, a menu titled 'UTILITIES' is open, showing various options. The 'UTILITIES' menu is highlighted in yellow. Below it, the 'DISTANCE' and 'ANGLE' options are also highlighted in yellow. The main window is black with white text. The text reads: 'Setting the modelling area with white background', 'UTILS', 'SNAPSHOT', 'Predefined colormaps', '2', and 'Evaluate DISTANCE, ANGLE,'. At the bottom right of the main window, there is a small 3D coordinate system with X, Y, and Z axes. The bottom of the interface features a toolbar with various icons and buttons. The 'UTILS' button is highlighted in yellow. The status bar at the bottom left shows 'Marc Mentat (C) Copyright 1994-2019, MSC Software Corporation, all rights reserved' and 'Command >'. The status bar at the bottom right shows 'Ready'.

UTILITIES

POSTSCRIPT SETTINGS

COLOR PRINT 1 GRAY PRINT 1

COLOR PRINT 2 GRAY PRINT 2

COLOR PRINT 3 GRAY PRINT 3

COLOR FILE GRAY FILE

SNAPSHOT ANIMATION

XDUMP 1 XDUMP 2

PARAMETERS ALIASES

PROCEDURES PYTHON

 NET MODULES

EDIT FILE LIST DIRECTORY

SYSTEM COMMAND SYSTEM SHELL

DISTANCE CALCULATE

ANGLE SAMPLE ELEMENT

SIZES

GENERALIZED XY PLOT

MORE

ALL SELEC VISIB OUTL TOP

EXIST UNSEL INVIS SURF BOT

SELECT SET END LIST (#)

RETURN MAIN

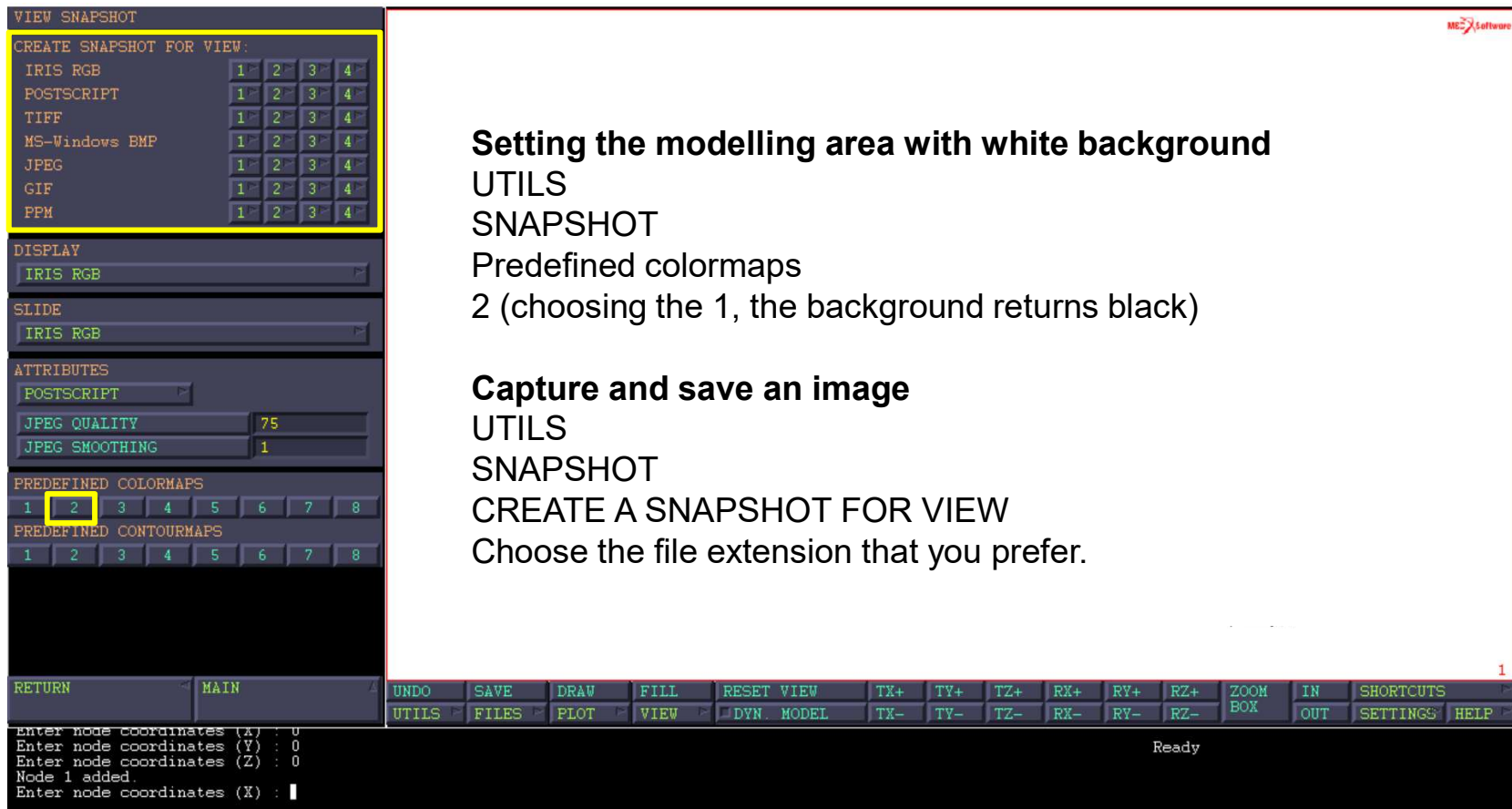
UNDO SAVE DRAW FILL RESET VIEW TX+ TY+ TZ+ RX+ RY+ RZ+ ZOOM IN SHORTCUTS

UTILS FILES PLOT VIEW DYN. MODEL TX- TY- TZ- RX- RY- RZ- BOX OUT SETTINGS HELP

Marc Mentat (C) Copyright 1994-2019, MSC Software Corporation, all rights reserved
Command > Ready

MSC Marc

TOOLBAR: UTILS



The screenshot shows the MSC Marc software interface. On the left, the UTILS menu is open, showing options for creating a snapshot for view (IRIS RGB, POSTSCRIPT, TIFF, MS-Windows BMP, JPEG, GIF, PPM) and displaying the current view (IRIS RGB). Below this, the SLIDE menu is also open, showing the current slide (IRIS RGB). The ATTRIBUTES menu is open, showing options for POSTSCRIPT, JPEG QUALITY (75), and JPEG SMOOTHING (1). The PREDEFINED COLORMAPS menu is open, showing a grid of buttons numbered 1 through 8, with button 2 highlighted. The PREDEFINED CONTOURMAPS menu is also open, showing a grid of buttons numbered 1 through 8. At the bottom, the toolbar is visible, with buttons for UNDO, SAVE, DRAW, FILL, RESET VIEW, TX+, TY+, TZ+, RX+, RY+, RZ+, ZOOM BOX, IN, SHORTCUTS, UTILS, FILES, PLOT, VIEW, DYN. MODEL, TX-, TY-, TZ-, RX-, RY-, RZ-, OUT, SETTINGS, and HELP. The status bar at the bottom right shows 'Ready'.

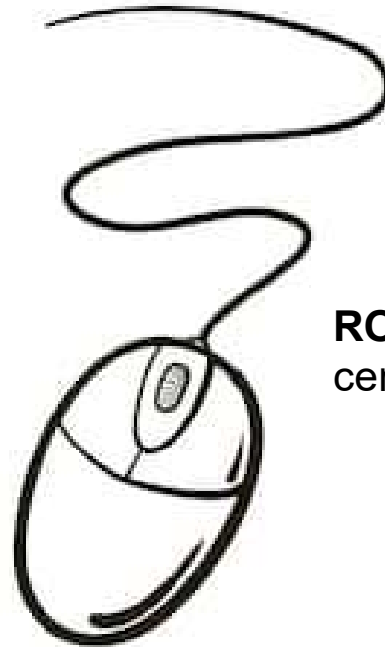
Setting the modelling area with white background
UTILS
SNAPSHOT
Predefined colormaps
2 (choosing the 1, the background returns black)

Capture and save an image
UTILS
SNAPSHOT
CREATE A SNAPSHOT FOR VIEW
Choose the file extension that you prefer.

MSC Marc

MOUSE GESTURE

ZOOM IN or ZOOM OUT
left button



ROTATE
center button

ORIZZONTAL or VERTICAL MOVEMENT
right button

When in the TOOLBAR
 DYNAMIC MODEL
is switched on.

Agenda

Finite Element Modelling

MSC Marc Mentat

Thin-walled profile in torsion

- The problem
- Mesh generation
- Geometric properties
- Material properties
- Links: RBE2
- Boundary conditions
- Jobs
- Results

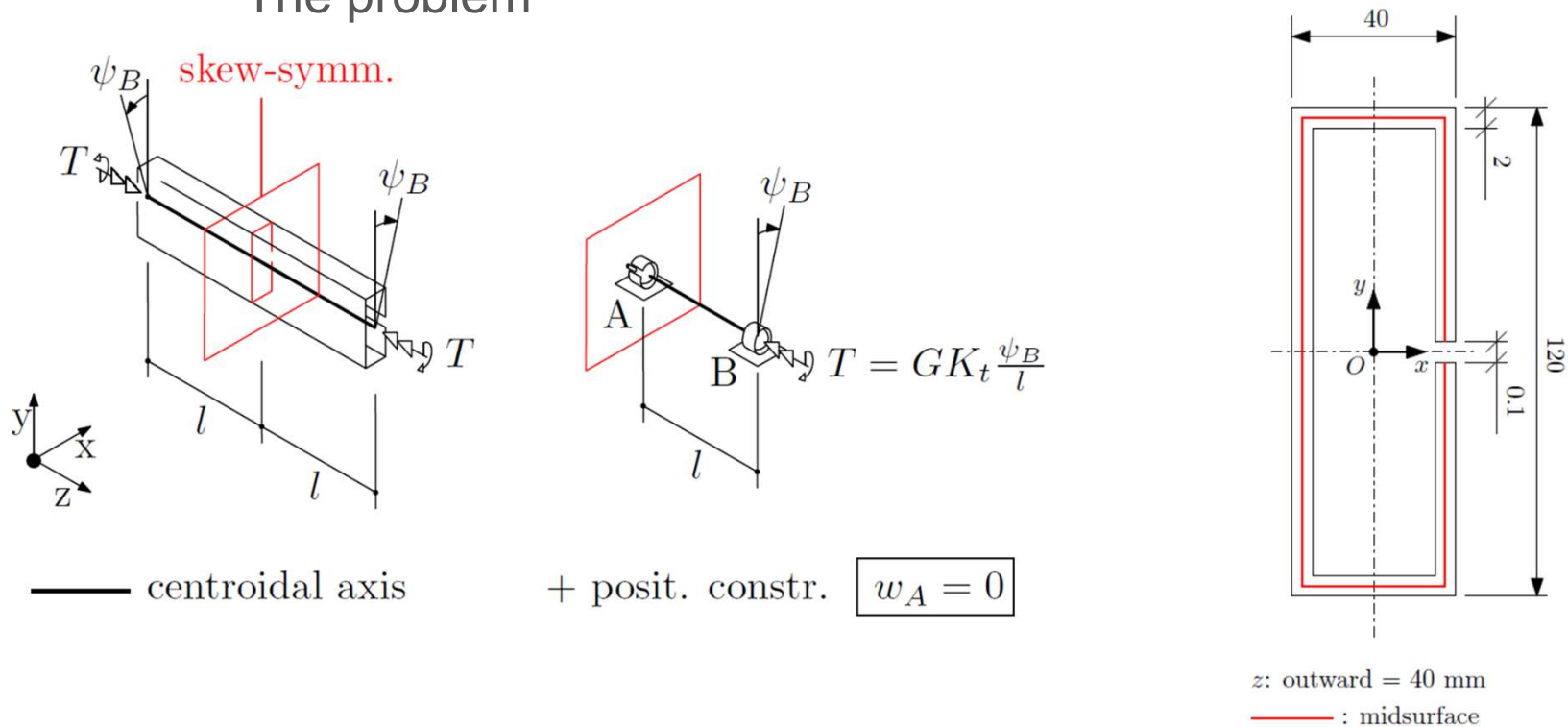
Thin-walled profile in torsion UPDATE

Mesh convergence

References

Thin-walled profile in torsion

The problem

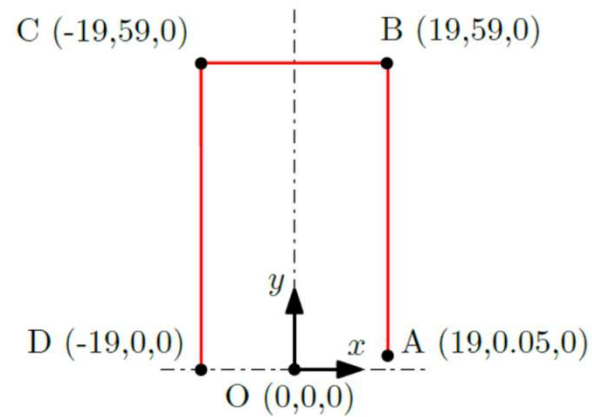
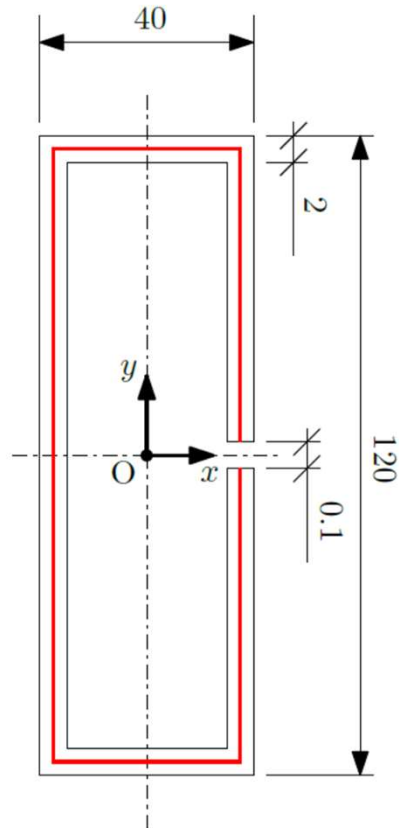


Open thin walled rectangular cross section profile where a crack is present at the lateral wall, a twist is imposed at the extremities of the profile.

Evaluate the torsional stiffness and the stress field occurring on the profile for the twist loading condition imposed by a rotation of 0.001 rad/mm, and the axial profile length (l) is equal to 40 mm.

Thin-walled profile in torsion

The problem

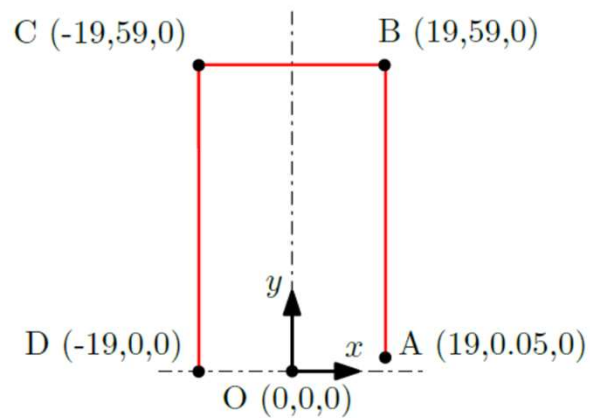


z : outward = 40 mm

— : midsurface

Thin-walled profile in torsion

MESH GENERATION



Definition of the crucial points that define the section

MESH GENERATION

NODES: ADD

0 0 0

19 0.05 0

19 59 0

-19 59 0

-19 0 0

RESET VIEW

FILL

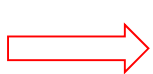
Thin-walled profile in torsion

MESH GENERATION

The screenshot displays a software interface for mesh generation. On the left, a vertical menu titled "MESH GENERATION" is highlighted with a red arrow. The menu items include: NODES, ELEMS, PTS, CRVS, SRFS, SOLIDS, BETWEEN NODE, BETWEEN POINT, ELEMENT CLASS (QUAD (4)), CURVE TYPE (LINE), SURFACE TYPE (QUAD), SOLID TYPE (BLOCK), COORDINATE SYSTEM (RECTANGULAR, GRID), CLEAR MESH, CLEAR GEOM, ATTACH, AUTOMESH, CHANGE CLASS, CHECK, CONVERT, DUPLICATE, ELEMENT TYPES, EXPAND, INTERSECT, MOVE, RELAX, RENUMBER, REVOLVE, SOLIDS, STRETCH, SUBDIVIDE, SWEEP, SYMMETRY, ALL, SELEC, VISIB, OUTL, TOP, EXIST, UNSEL, INVIS, SURF, BOT, SELECT, SET, END LIST (#), RETURN, MAIN. The main window shows a 3D coordinate system with X, Y, and Z axes. The bottom status bar shows "Ready".

Thin-walled profile in torsion

MESH GENERATION



ADD: This command adds a *node, element, point, curve, surface, solid* at the specified location.

REM: This command removes nodes from the model. You must specify a list of nodes.

EDIT: This command allows you to relocate an existing node. You must specify *node, element, point, curve, surface, solid* and its new coordinates.

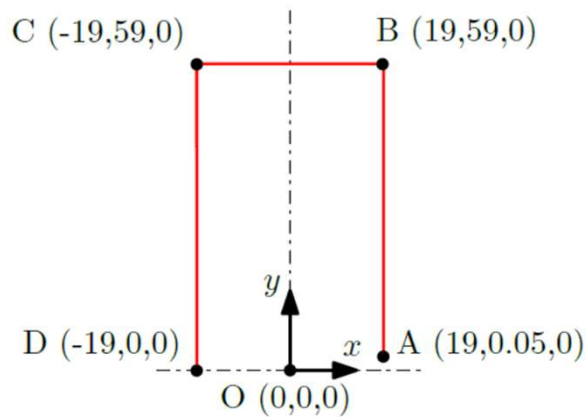
SHOW: This command displays information about the specified *node, element, point, curve, surface, solid* at the prompt command.

Thin-walled profile in torsion

MESH GENERATION

Midsurface definition by 1D elements.

Any element is defined between two nodes, in this model three curves are adopted: AB, BC, CD.



MESH GENERATION

ELEMENT CLASS: LINE (2)

CURVES: ADD

Select node A

Select node B

Select node B

Select node C

Select node C

Select node D

RESET VIEW

FILL

Thin-walled profile in torsion

MESH GENERATION

The screenshot displays the MESH GENERATION menu on the left and a 2D wireframe model of a thin-walled profile on the right. The menu includes options for adding, removing, editing, and showing various mesh elements (Nodes, Elements, Points, Curves, Surfaces, Solids). It also includes options for setting the element class (QUAD (4)), curve type (LINE), surface type (QUAD), and solid type (BLOCK). The coordinate system is set to RECTANGULAR. The 2D model shows a rectangular profile with a central vertical slot, defined by red lines. A coordinate system with X and Y axes is visible in the bottom right corner of the model area. The software interface includes a command line at the bottom left and a toolbar at the bottom right.

Command > *set_curve_type line
Command > *set_curve_type circle_cr
Command > *set_curve_type line
Command > *add_curves
Enter line points : █

Ready

Thin-walled profile in torsion

MESH GENERATION

Nodes labelling check:

PLOT
NODES: SETTING
 LABELS
REGEN

```
Command > *set_curve_type line
Command > *add_curves
Enter line points : *set_node_labels on
Command > *regenerate
Command > |
```


Thin-walled profile in torsion

MESH GENERATION

The screenshot displays the 'NODE PLOT SETTINGS' dialog box on the left, with a red arrow pointing to the 'DRAW' section. The 'DRAW' section includes checkboxes for 'NODES', 'LABELS', and 'COORDS/TRANSFORMS INFO'. Below it, the 'RELATED PLOT SETTINGS' section has a 'TRANSFORMATION' dropdown menu. At the bottom of the dialog are buttons for 'RESET', 'DRAW', 'REDRAW', and 'REGEN'. The main plot area on the right shows a rectangular profile with nodes labeled 1 through 5. A red arrow points to node 5. The bottom of the interface features a command line with the following text: 'Command > *set_curve_type line', 'Command > *add_curves', 'Enter line points : *set_node_labels on', 'Command > *regenerate', and 'Command > |'. A toolbar at the bottom includes buttons for 'UNDO', 'SAVE', 'DRAW', 'FILL', 'RESET VIEW', 'TX+', 'TY+', 'TZ+', 'RX+', 'RV+', 'RZ+', 'ZOOM BOX', 'IN', 'SHORTCUTS', 'UTILS', 'FILES', 'PLOT', 'VIEW', 'DYN. MODEL', 'TX-', 'TY-', 'TZ-', 'RX-', 'RV-', 'RZ-', 'OUT', 'SETTINGS', and 'HELP'. The 'PLOT' button is highlighted with a yellow box. The status bar at the bottom right shows 'Ready'.

Thin-walled profile in torsion

MESH GENERATION



CURVE PLOT SETTINGS

DRAW

- CURVES
- DIRECTION
- LABELS

CURVE FACETTING

◆ RELATIVE ◀ ABSOLUTE

TOLERANCE 0.1

MIN DEPTH 1

MAX DEPTH 7

PREDEFINED SETTINGS

LOW MEDIUM HIGH

ARROW PLOT SETTINGS

RESET DRAW REDRAW REGEN

1

RETURN MAIN UNDO SAVE DRAW FILL RESET VIEW TX+ TY+ TZ+ RX+ RY+ RZ+ ZOOM BOX IN SHORTCUTS

UTILS FILES PLOT VIEW DYN. MODEL TX- TY- TZ- RX- RV- RZ- OUT SETTINGS HELP

Command > *regenerate
Command > *set_curve_direction on
Command > *set_curve_labels on
Command > *regenerate
Command > |

Ready

Curves labelling check:

PLOT
CURVES: SETTING
 LABELS
 DIRECTIONS
REGEN

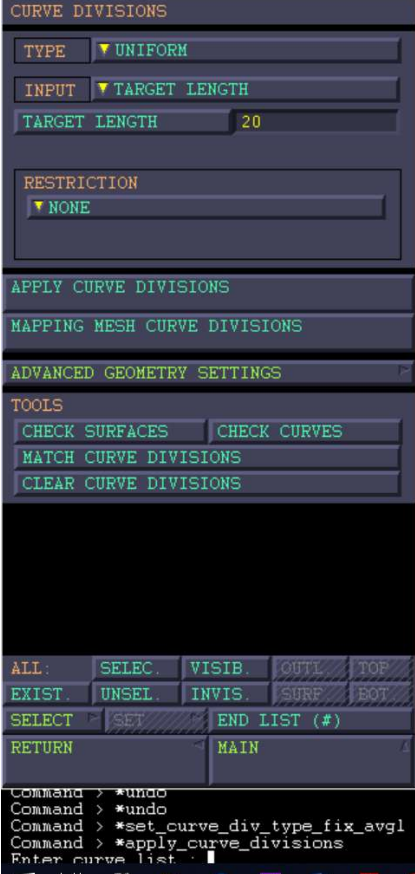
Thin-walled profile in torsion

MESH GENERATION

The image displays a CAD software interface with a settings panel on the left and a 3D model on the right. The settings panel is titled "CURVE PLOT SETTINGS" and includes sections for "DRAW", "CURVE FACETTING", and "ARROW PLOT SETTINGS". A red arrow points to the "DRAW" section, which contains expandable options for "CURVES", "DIRECTION", and "LABELS". The "CURVE FACETTING" section shows "RELATIVE" selected over "ABSOLUTE", with "TOLERANCE" set to 0.1, "MIN DEPTH" at 1, and "MAX DEPTH" at 7. The "ARROW PLOT SETTINGS" section has "LOW", "MEDIUM", and "HIGH" options. The 3D model shows a thin-walled profile with a red mesh. A red arrow points to the mesh on the left side of the profile. The model is labeled with numbers 1, 2, 3, 4, and 5 at various points. A coordinate system with X, Y, and Z axes is visible in the bottom right corner. The software interface includes a command line at the bottom with the following text: "Command > *regenerate", "Command > *set_curve_direction on", "Command > *set_curve_labels on", "Command > *regenerate", and "Command > |". The status bar at the bottom right shows "Ready".

Thin-walled profile in torsion

MESH GENERATION



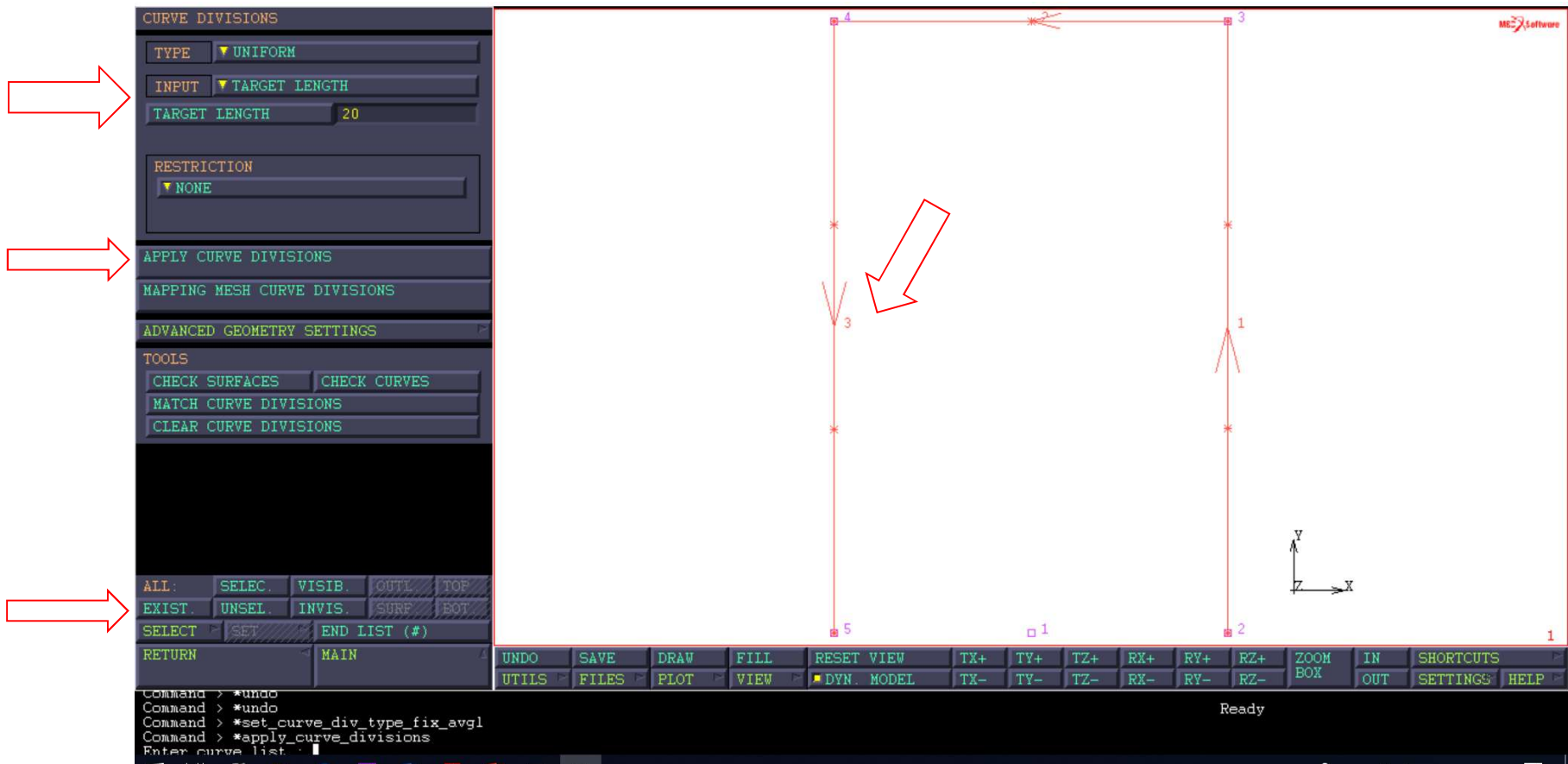
Curve division:

MESH GENERATION
AUTOMESH
PRELIMINARY: CURVE DIVISIONS
TYPE: UNIFORM
INPUT: TARGET LENGTH
TARGET LENGTH: 20
APPLY CURVE DIVISIONS
ALL: EXIST

The curve division is defined by the engineering practice, and it is strictly related to the elements mesh size.

Thin-walled profile in torsion

MESH GENERATION



Thin-walled profile in torsion

MESH GENERATION

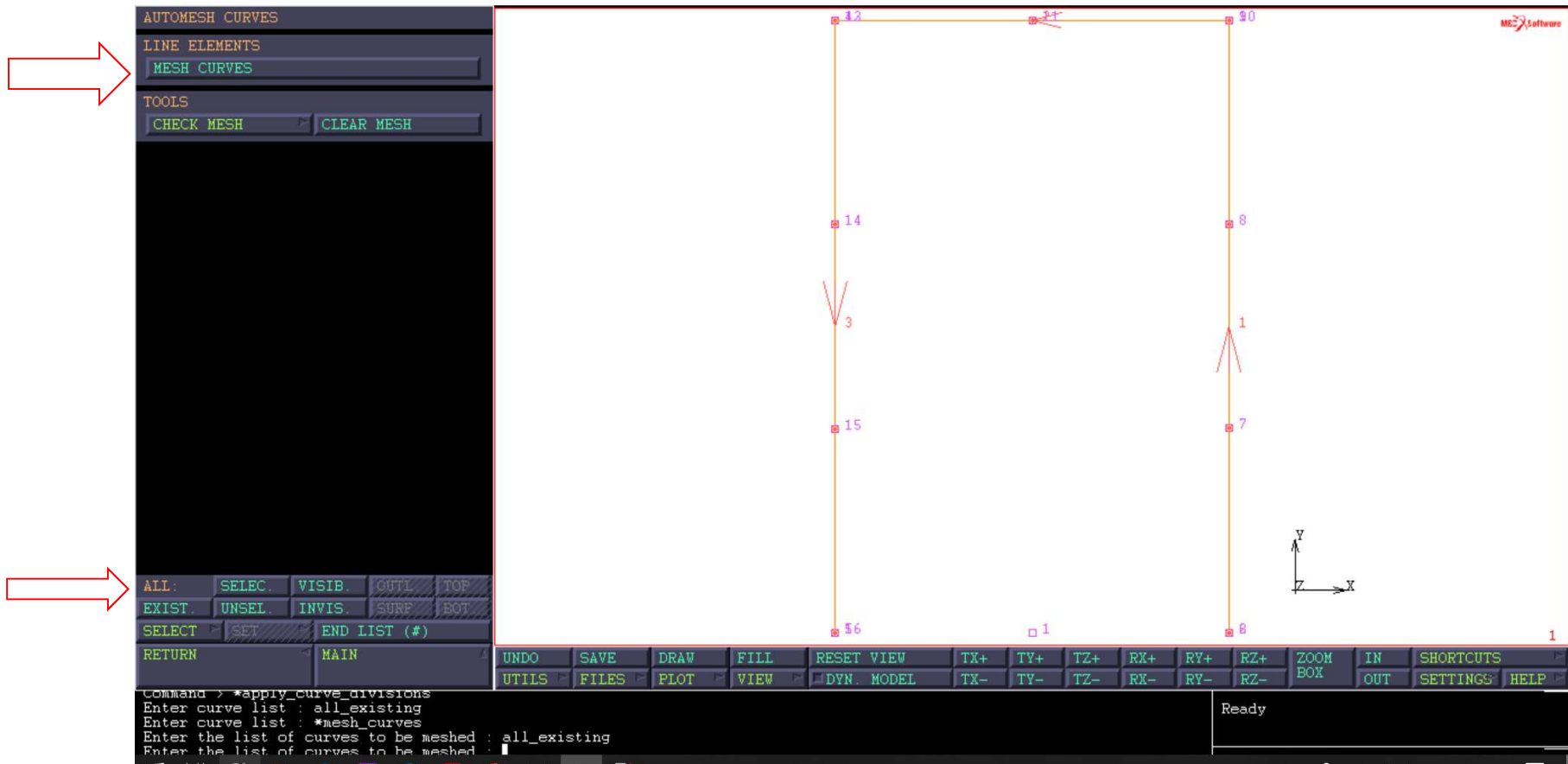
From curves to elements:

MESH GENERATION
AUTOMESH
PRELIMINARY: CURVE DIVISIONS
CHOOSE: CURVE MESHING
LINE ELEMS: MESH CURVES
ALL: EXIST

Command > *apply_curve_divisions
Enter curve list : all_existing
Enter curve list : *mesh_curves
Enter the list of curves to be meshed : all_existing
Enter the list of curves to be meshed :

Thin-walled profile in torsion

MESH GENERATION



Thin-walled profile in torsion

MESH GENERATION



From 1D elements to shell elements

MESH GENERATION

EXPAND

TRANSLATIONS:

0 0 20

REPETITIONS

2

MODE: REMOVE

ELEMENTS

ALL:EXIST

The elements size is defined by the engineering practice, and it is related to average size of the component under scrutiny. In the present model, along the Z axis, we decide to adopt two elements to maintain the elements almost squared.

Thin-walled profile in torsion

MESH GENERATION

The screenshot displays the software interface for mesh generation. On the left, a command window shows the following commands:

```
Command > *expand_remove  
Command > *expand_elements  
Enter expand element list : all_existing  
Enter expand element list : *dynamic_model_on  
Enter expand element list : |
```

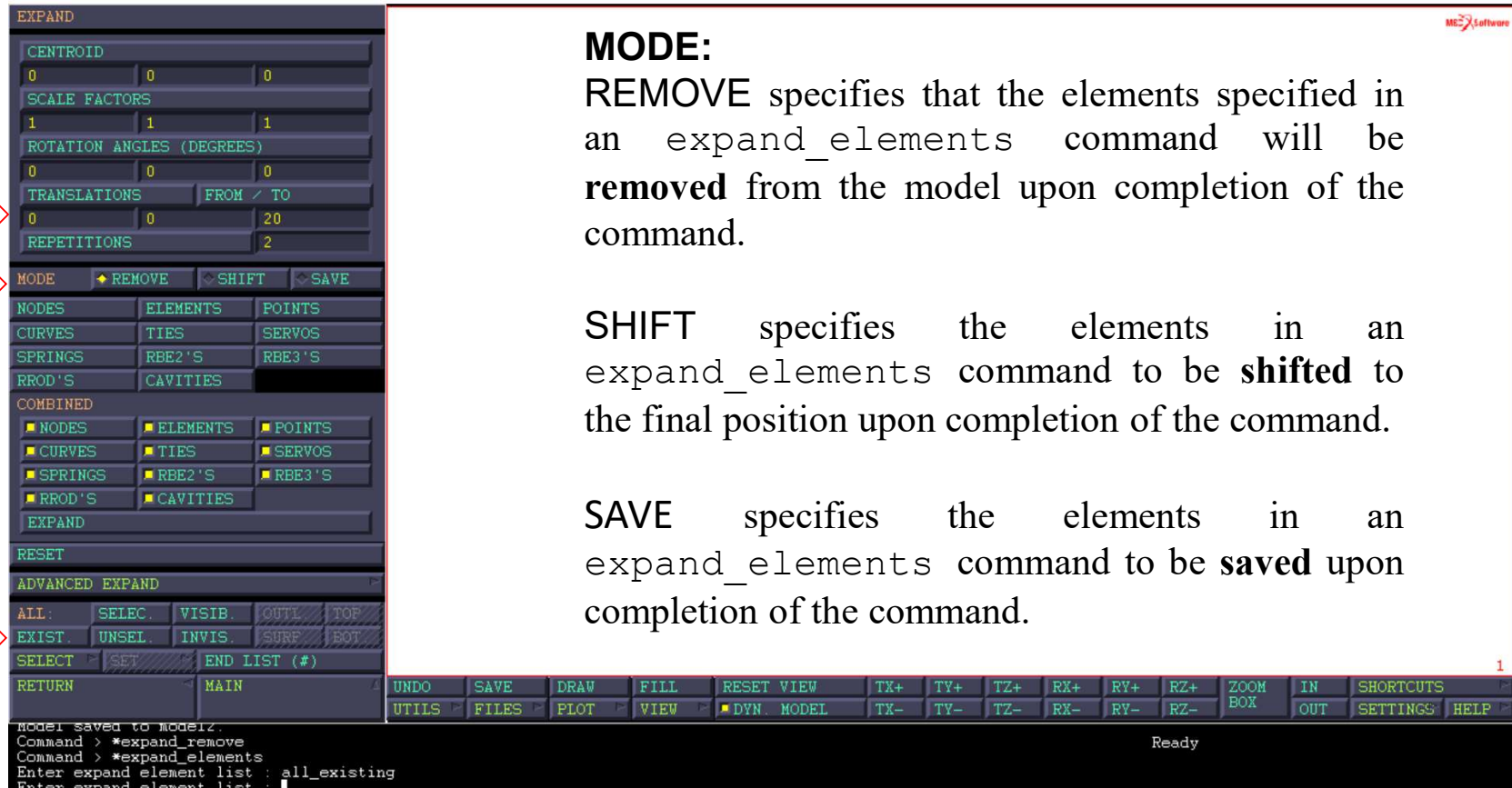
The main interface features a menu on the left with the following sections:

- EXPAND**
 - CENTROID: 0 0 0
 - SCALE FACTORS: 1 1 1
 - ROTATION ANGLES (DEGREES): 0 0 0
 - TRANSLATIONS: FROM / TO (0 0 20)
 - REPETITIONS: 2
- MODE**: REMOVE, SHIFT, SAVE
- NODES**, **ELEMENTS**, **POINTS**
- CURVES**, **TIES**, **SERVOS**
- SPRINGS**, **RBE2'S**, **RBE3'S**
- RROD'S**, **CAVITIES**
- COMBINED**
 - NODES, ELEMENTS, POINTS
 - CURVES, TIES, SERVOS
 - SPRINGS, RBE2'S, RBE3'S
 - RROD'S, CAVITIES
- EXPAND
- RESET
- ADVANCED EXPAND**
 - ALL: SELEC, VISIB, OUTL, TOP
 - EXIST, UNSEL, INVIS, SURF, BOT
 - SELECT, SET, END LIST (#)
- RETURN, MAIN

The 3D model on the right shows a thin-walled profile with a mesh of elements. The profile is a rectangular cross-section with a central hole. The mesh is composed of quadrilateral elements. The nodes are numbered from 1 to 71. The profile is shown in a perspective view, with a coordinate system (X, Y, Z) visible in the bottom right corner. The software logo "MSC Software" is in the top right corner.

Thin-walled profile in torsion

MESH GENERATION



MODE:

REMOVE specifies that the elements specified in an `expand_elements` command will be **removed** from the model upon completion of the command.

SHIFT specifies the elements in an `expand_elements` command to be **shifted** to the final position upon completion of the command.

SAVE specifies the elements in an `expand_elements` command to be **saved** upon completion of the command.

Thin-walled profile in torsion

MESH GENERATION



For geometry view-point, the rectangular open profile is symmetric respect to xz plane with normal vector the y axis.

MESH GENERATION

SYMMETRY

SYMMETRY PLANE

POINT

0 0 0

NORMAL

0 1 0

ELEMENTS

ALL:EXIST

ME software

martedì 26 marzo 2019

Thin-walled profile in torsion

MESH GENERATION

SYMMETRY

SYMMETRY PLANE

POINT	0	0	0
NORMAL	FROM / TO		
1	0	0	0

CREATE NEW MATCHING BOUND'S

NODES	ELEMENTS	POINTS
CURVES	SURFACES	SOLIDS
TIES	SERVOS	SPRINGS
RBE2'S	RBE3'S	RROD'S

COMBINED

<input type="checkbox"/> NODES	<input type="checkbox"/> ELEMENTS	<input type="checkbox"/> POINTS
<input type="checkbox"/> CURVES	<input type="checkbox"/> SURFACES	<input type="checkbox"/> SOLIDS
<input type="checkbox"/> TIES	<input type="checkbox"/> SERVOS	<input type="checkbox"/> SPRINGS
<input type="checkbox"/> RBE2'S	<input type="checkbox"/> RBE3'S	<input type="checkbox"/> RROD'S
<input type="checkbox"/> CAVITIES		

SYMMETRY

RESET

ALL:	SELEC.	VISIB.	OUTL	TOP
EXIST.	UNSEL.	INVIS.	SURF	BOT

SELECT SET END LIST (#)

RETURN MAIN

UNDO SAVE DRAW FILL RESET VIEW TX+ TY+ TZ+ RX+ RV+ RZ+ ZOOM IN SHORTCUTS
UTILS FILES PLOT VIEW DYN. MODEL TX- TY- TZ- RX- RY- RZ- BOX OUT SETTINGS HELP

Enter symmetry plane normal : 0 1 0
Command > *symmetry_elements
Enter symmetry element list : all_existing
Enter symmetry element list : *symmetry_reset
Enter symmetry element list :

Ready

martedì 26 marzo 2019

The lateral crack is difficult to be detected. However, either by adopting the node labelling plot, or by a zoom box it might be easily checked.

Thin-walled profile in torsion

MESH GENERATION

SYMMETRY

SYMMETRY PLANE

POINT	0	0	0
NORMAL	FROM / TO		
1	0	0	0

CREATE NEW MATCHING BOUND'S

NODES	ELEMENTS	POINTS
CURVES	SURFACES	SOLIDS
TIES	SERVOS	SPRINGS
RBE2'S	RBE3'S	RROD'S

COMBINED

<input checked="" type="checkbox"/> NODES	<input checked="" type="checkbox"/> ELEMENTS	<input checked="" type="checkbox"/> POINTS
<input checked="" type="checkbox"/> CURVES	<input checked="" type="checkbox"/> SURFACES	<input checked="" type="checkbox"/> SOLIDS
<input checked="" type="checkbox"/> TIES	<input checked="" type="checkbox"/> SERVOS	<input checked="" type="checkbox"/> SPRINGS
<input checked="" type="checkbox"/> RBE2'S	<input checked="" type="checkbox"/> RBE3'S	<input checked="" type="checkbox"/> RROD'S
<input checked="" type="checkbox"/> CAVITIES		

SYMMETRY

RESET

ALL:	SELEC.	VISIB.	OUTL	TOP
EXIST.	UNSEL.	INVIS.	SURF	BOT
SELECT	SET	END LIST (#)		

RETURN MAIN

UNDO SAVE DRAW FILL RESET VIEW TX+ TY+ TZ+ RX+ RV+ RZ+ ZOOM IN SHORTCUTS
UTILS FILES PLOT VIEW DYN. MODEL TX- TY- TZ- RX- RY- RZ- BOX OUT SETTINGS HELP

Enter symmetry plane normal : 0 1 0
Command > *symmetry_elements
Enter symmetry element list : all_existing
Enter symmetry element list : *symmetry_reset
Enter symmetry element list :

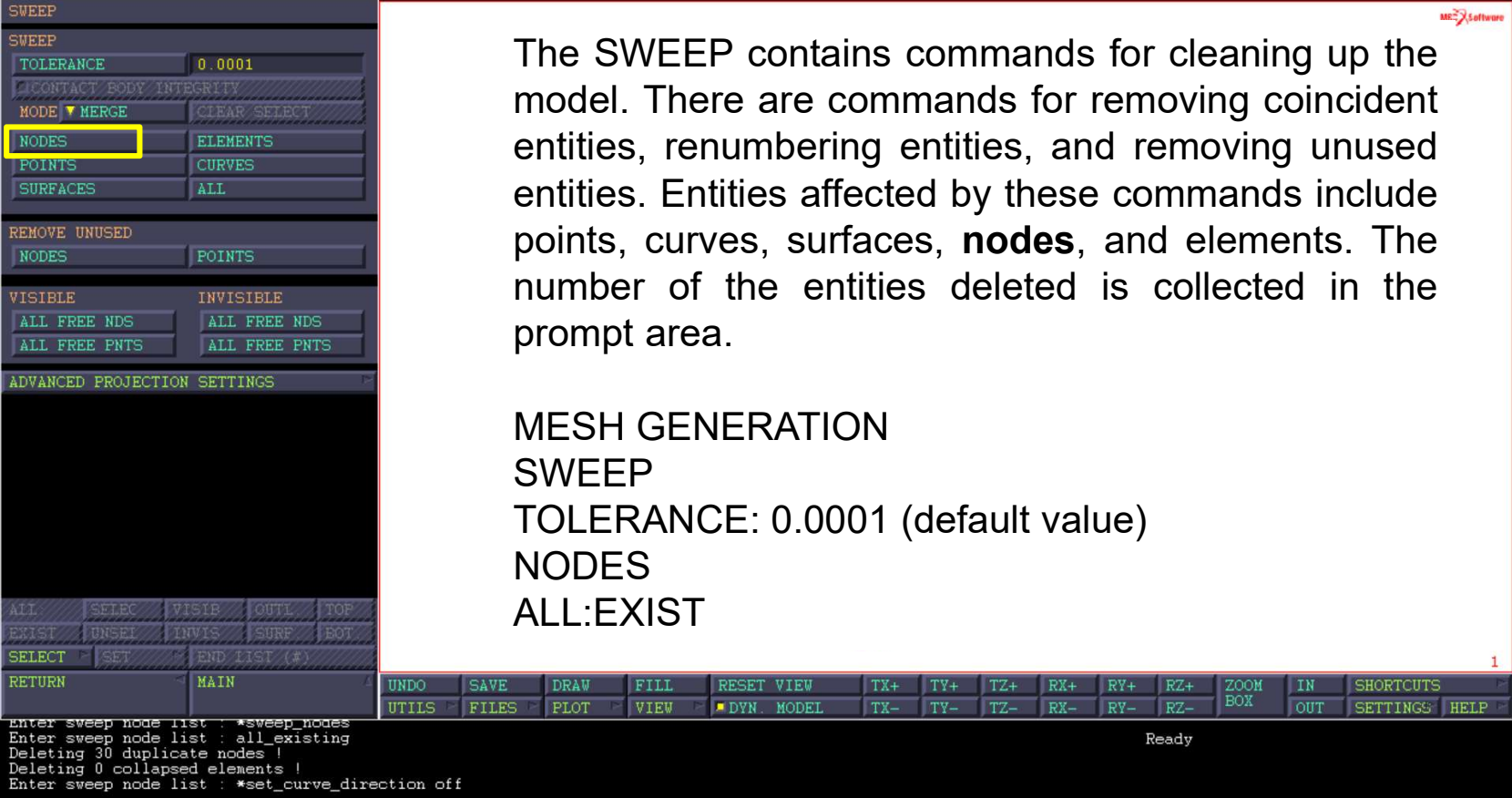
Ready

martedì 26 marzo 2019

Zoom box

Thin-walled profile in torsion

MESH GENERATION



The SWEEP contains commands for cleaning up the model. There are commands for removing coincident entities, renumbering entities, and removing unused entities. Entities affected by these commands include points, curves, surfaces, **nodes**, and elements. The number of the entities deleted is collected in the prompt area.

MESH GENERATION
SWEEP
TOLERANCE: 0.0001 (default value)
NODES
ALL:EXIST

```
Enter sweep node list : *sweep_nodes
Enter sweep node list : all_existing
Deleting 30 duplicate nodes !
Deleting 0 collapsed elements !
Enter sweep node list : *set_curve_direction off
```


Thin-walled profile in torsion

MESH GENERATION

SWEEP

SWEEP

TOLERANCE 0.0001

CONTACT BODY INTEGRITY

MODE ▾ MERGE CLEAR SELECT

NODES ELEMENTS

POINTS CURVES

SURFACES ALL

REMOVE UNUSED

NODES POINTS

VISIBLE INVISIBLE

ALL FREE NDS ALL FREE NDS

ALL FREE PNTS ALL FREE PNTS

ADVANCED PROJECTION SETTINGS

ALL SELEC VISIB OUTL TOP

EXIST UNSEL INVIS SURF BOT

SELECT SET END LIST (#)

RETURN MAIN

UNDO SAVE DRAW FILL RESET VIEW TX+ TY+ TZ+ RX+ RV+ RZ+ ZOOM IN SHORTCUTS

UTILS FILES PLOT VIEW DYN. MODEL TX- TY- TZ- RX- RV- RZ- ZOOM BOX OUT SETTINGS HELP

Enter sweep node list : *sweep_nodes
Enter sweep node list : all_existing
Deleting 30 duplicate nodes !
Deleting 0 collapsed elements !
Enter sweep node list : *set_curve_direction off

Ready

Thin-walled profile in torsion

MESH GENERATION

MCX software

Deleting 0 duplicate nodes !
Deleting 0 collapsed elements !
Enter sweep node list : *set_node_labels on
Command > *regenerate
Command > |

Ready

1 nuova notifica

The nodes located at a distance that overcome the tolerance threshold are deleted, the further remain inalterated. Therefore, the lateral crack remains.

Agenda

Finite Element Modelling

MSC Marc Mentat

Thin-walled profile in torsion

- The problem
- Mesh generation
- **Geometric properties**
- Material properties
- Links: RBE2
- Boundary conditions
- Jobs
- Results

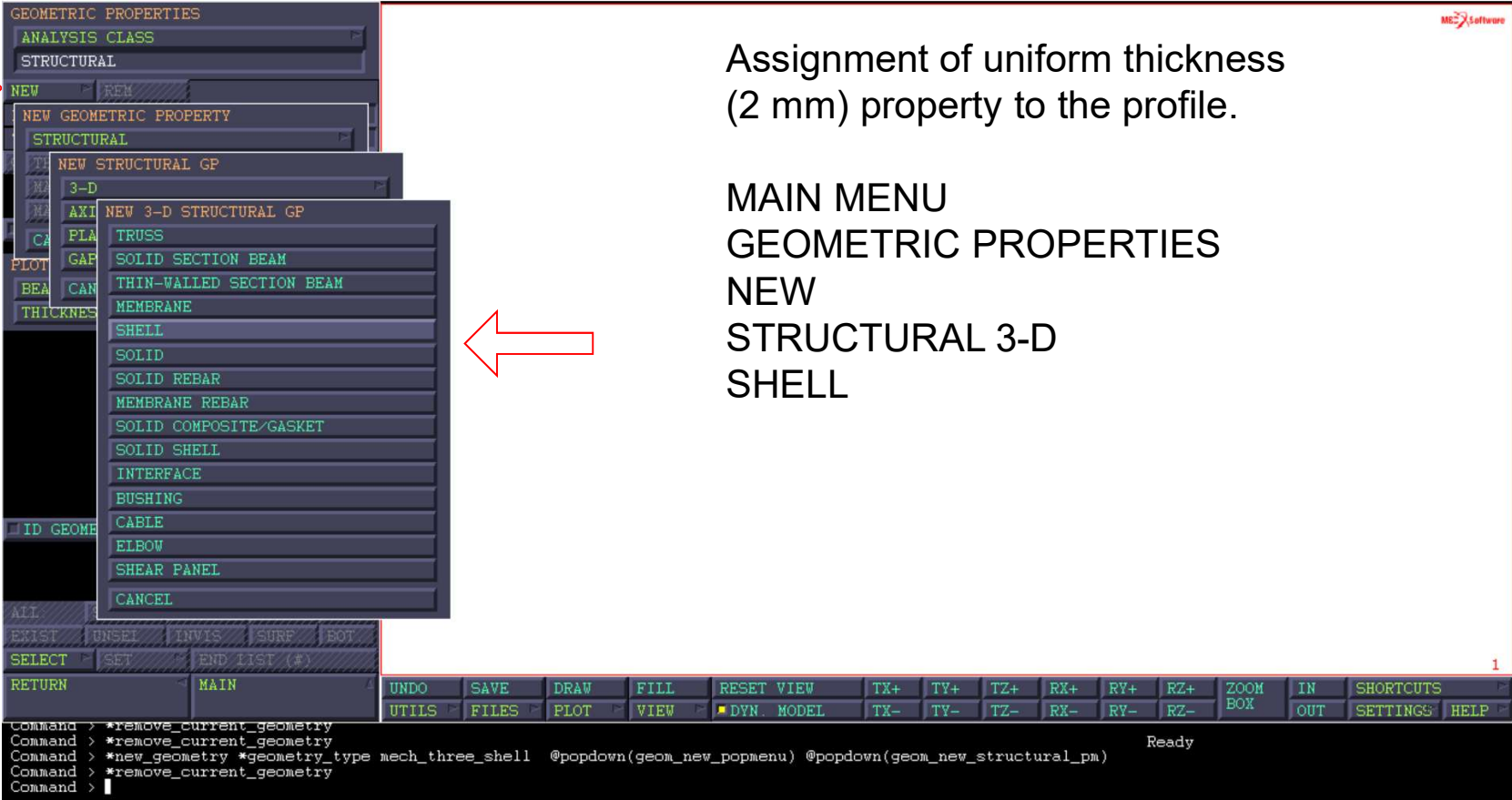
Thin-walled profile in torsion UPDATE

Mesh convergence

References

Thin-walled profile in torsion

GEOMETRIC PROPERTIES



Assignment of uniform thickness (2 mm) property to the profile.

MAIN MENU
GEOMETRIC PROPERTIES
NEW
STRUCTURAL 3-D
SHELL

Command > *remove_current_geometry
Command > *remove_current_geometry
Command > *new_geometry *geometry_type mech_three_shell @popdown(geom_new_popmenu) @popdown(geom_new_structural_pm)
Command > *remove_current_geometry
Command >

Thin-walled profile in torsion

GEOMETRIC PROPERTIES

Assignment of uniform thickness (2 mm) property to the profile.

NAME
thickness_2mm (optional)
PROPERTIES
THICKNESS: 2
OK

Thin-walled profile in torsion

GEOMETRIC PROPERTIES

Assignment of the uniform thickness (2 mm) property to the profile.

ELEMENTS: ADD
EXIST

```
Command > *geometry_param thick
Enter geometric property parameter value : 2
Command > *add_geometry_elements
Enter geometry add element list : all_existing
Enter geometry add element list :
```

Thin-walled profile in torsion

GEOMETRIC PROPERTIES

The screenshot displays the 'GEOMETRIC PROPERTIES' panel on the left side of the software interface. The panel is divided into several sections:

- ANALYSIS CLASS:** STRUCTURAL
- NEW:** REM
- NAME:** thickness_2mm
- TYPE:** mech_three_shell
- COPY:** PREV, NEXT, EDIT
- PROPERTIES:** BEAM SECTIONS, TABLES
- PLOT SETTINGS:** BEAM, SHELL, THICKNESS DIRECTION
- ID GEOMETRIES:** TOOLS
- ELEMENTS:** ADD, REM, 32
- SURFACES:** ADD, REM, 0
- ALL:** SELEC, VISIB, OUTL, TOP
- EXIST.:** INSEL, INVIS, SURF, BOT
- SELECT:** SET, END LIST (#)
- RETURN:** MAIN

The 3D model on the right shows a rectangular thin-walled profile with blue arrows indicating torsion. A coordinate system (X, Y, Z) is visible in the bottom right corner of the model area.

Command > *geometry_param thick
Enter geometric property parameter value : 2
Command > *add_geometry_elements
Enter geometry add element list : all_existing
Enter geometry add element list :

Thin-walled profile in torsion

GEOMETRIC PROPERTIES

GEOMETRIC PROPERTIES

ANALYSIS CLASS
STRUCTURAL

NEW REM
NAME thickness_2mm
TYPE mech_three_shell
COPY PREV NEXT EDIT

PROPERTIES

BEAM SECTIONS TABLES

PLOT SETTINGS
BEAM SHELL
THICKNESS DIRECTION

thickness_2mm

Visual check to
geometric properties
definition

ID GEOMETRY

1

Ready

```
Command > *identity_backfaces *regen
Command > *draw_shells_expanded off
Command > *draw_offsets off
Command > *identify_geometries *regen
Command > |
```


Thin-walled profile in torsion

GEOMETRIC PROPERTIES

thickness_2mm

Moving to the menu
PLOT SETTING:
SHELL

1

Ready

Thin-walled profile in torsion

GEOMETRIC PROPERTIES

3D visual check of geometric properties

thickness_2mm

- PLOT EXPANDED
- PLOT OFFSET
- SOLID

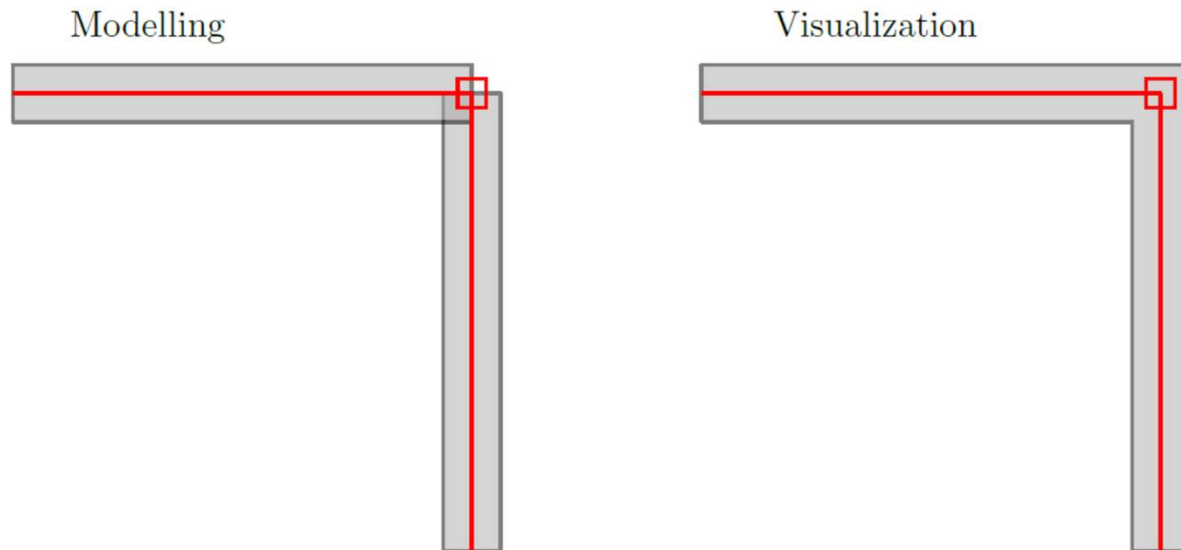
The thickness is equally distributed from the midshell

Command > *reset_view
Command > *fill_view
Command > *identify_none *regen
Command > *identify_geometries *regen

Ready

Thin-walled profile in torsion

GEOMETRIC PROPERTIES



At the corner, the visualization differs from the modelling aspect. In fact, the shell thickness remains normal to the elements located at the mid-plane, therefore the visualization is not coherent with the modelling aspect.

The mid-plane is a reference plane commonly adopted for the evaluation of the stress and strain fields of a shell element during FE analysis. However, ...

Thin-walled profile in torsion

GEOMETRIC PROPERTIES

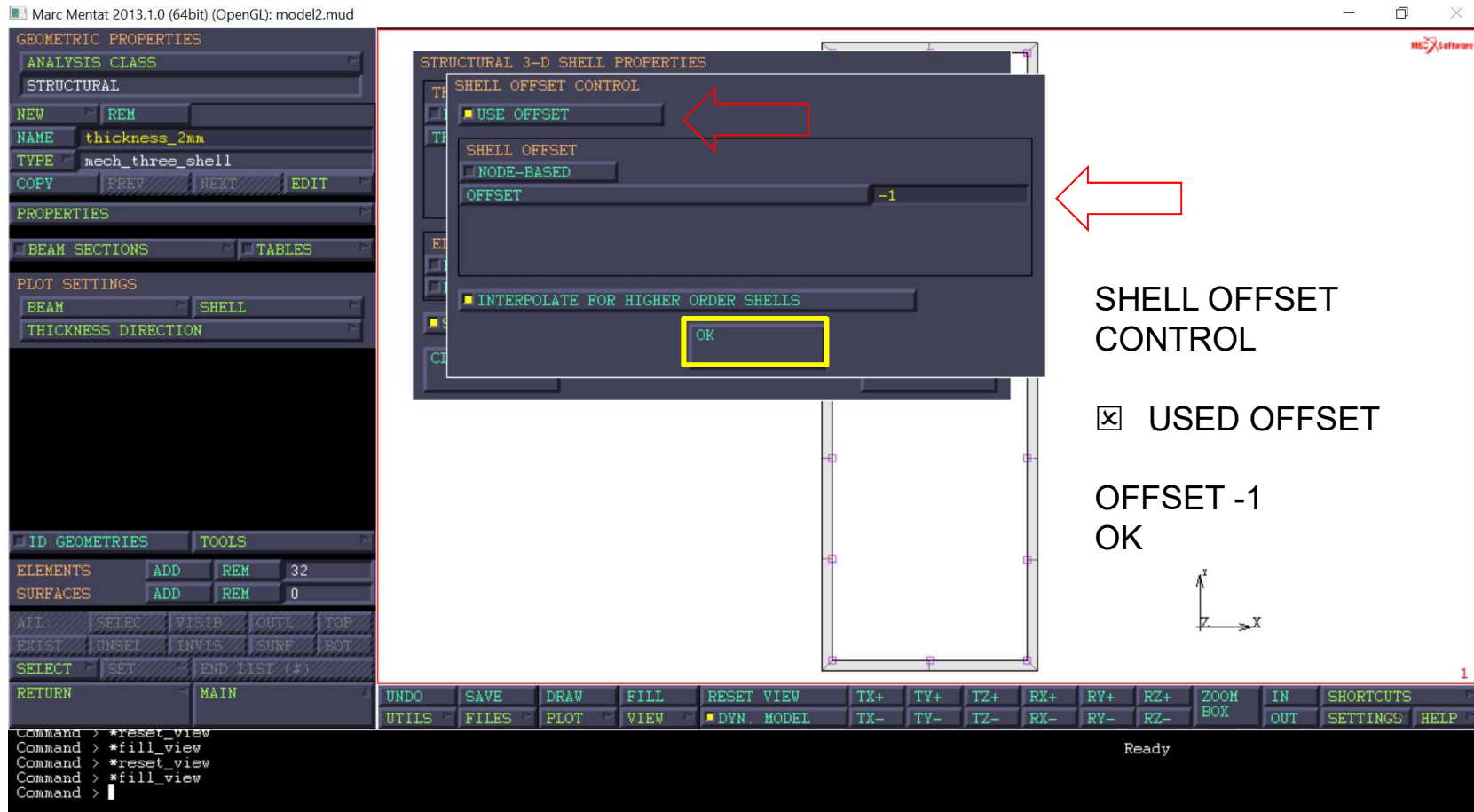
However, we can modify the reference plane used for the output evaluation by imposing a offset.

The screenshot displays the ANSYS software interface. On the left, the 'GEOMETRIC PROPERTIES' panel is visible, with the 'PROPERTIES' tab selected. The 'STRUCTURAL 3-D SHELL PROPERTIES' dialog box is open in the center, showing the 'THICKNESS' set to 2 and the 'SHELL OFFSET' checkbox checked. The 'ELEMENT TECHNOLOGY' section includes 'MEMBRANE ONLY' and 'FLAT ELEMENT (TYPE 49 ONLY)'. The 'CLEAR' and 'OK' buttons are at the bottom of the dialog. On the right, the text 'PROPERTIES' and '☒ SHELL OFFSET' is displayed. The bottom of the screen shows the command line with the following commands: `*fill view`, `*identify_none *regen`, `*clear_geometry_values`, `*geometry_param thick 2`, and a cursor. The status bar at the bottom right indicates 'Ready'.

Thin-walled profile in torsion

GEOMETRIC PROPERTIES

However, we can modify the reference plane by imposing a offset.

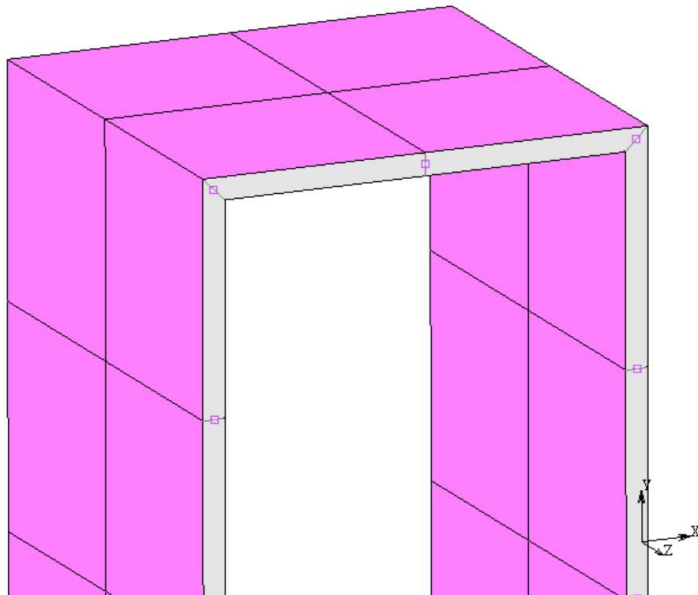


Thin-walled profile in torsion

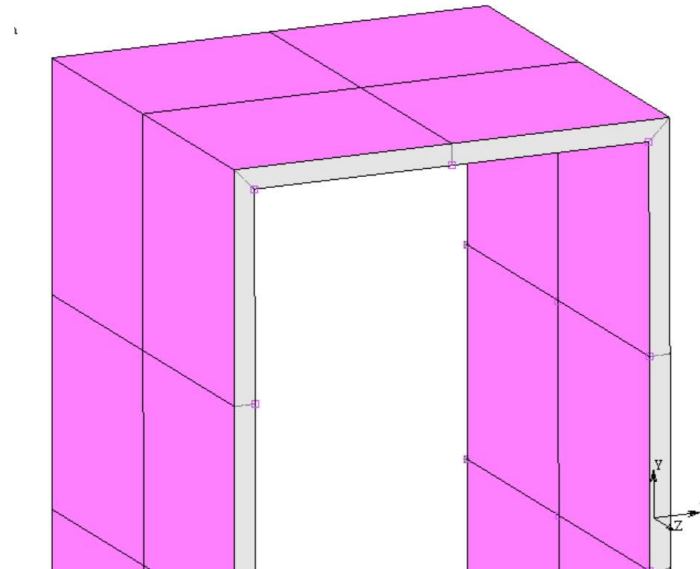
GEOMETRIC PROPERTIES

However, we can modify the reference plane by imposing a offset, calculated from the normal vector of each element, and with the magnitude prescribed by the designer.

OFFSET = 0

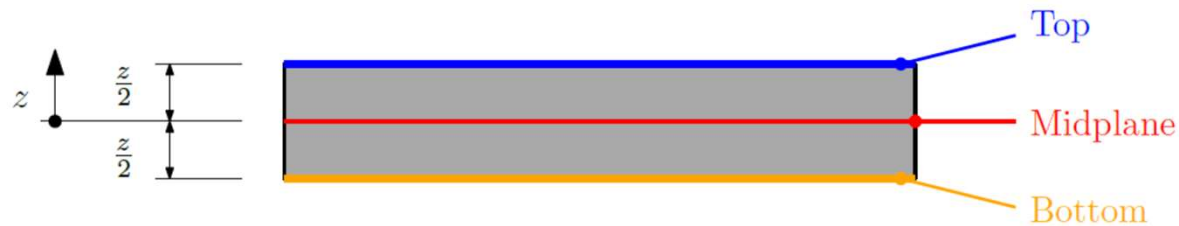


OFFSET = -1



Thin-walled profile in torsion

GEOMETRIC PROPERTIES



In the present treatise, the reference surface is assumed to be the plane where the nodes lie.

An *offset* term is considered that pointwisely shifts the geometric midsurface with respect to the reference surface.

A positive *offset* shifts the midsurface towards the top.

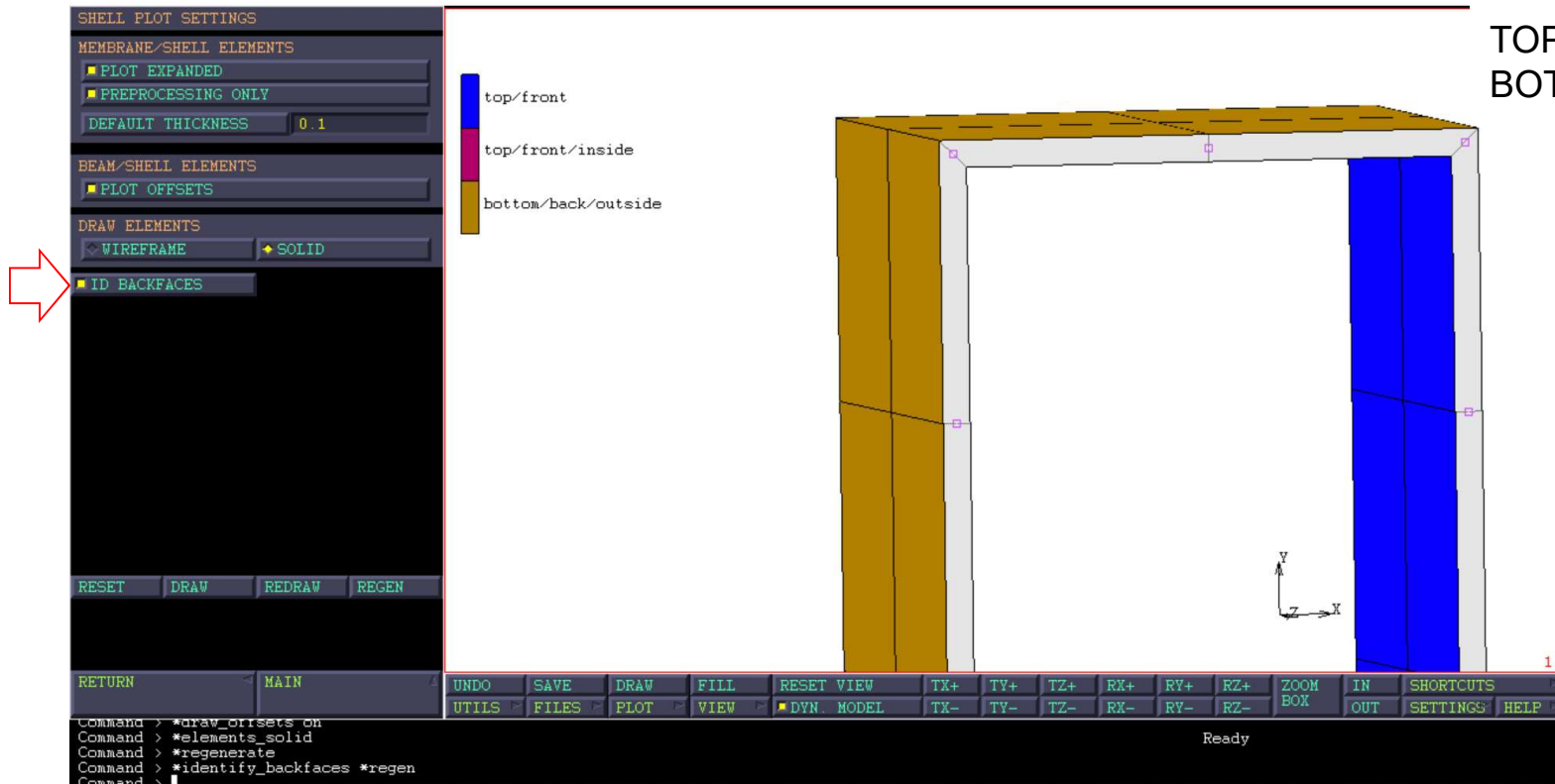
Thin-walled profile in torsion

GEOMETRIC PROPERTIES

Check of bottom and top surfaces.

ID BACKFACES

TOP: blue
BOTTOM: ochre



Thin-walled profile in torsion

GEOMETRIC PROPERTIES

MESH GENERATION
CHECK
FLIP ELEMENTS
ALL: EXIST

top/front
top/front/inside
bottom/back/outside

Command > *rlll_view
Command > *geometry_option offsets:off
Command > *flip_elements
Enter flip element list : all_existing
Enter flip element list :

Ready

Agenda

Finite Element Modelling

MSC Marc Mentat

Thin-walled profile in torsion

- The problem
- Mesh generation
- Geometric properties
- **Material properties**
- Links: RBE2
- Boundary conditions
- Jobs
- Results

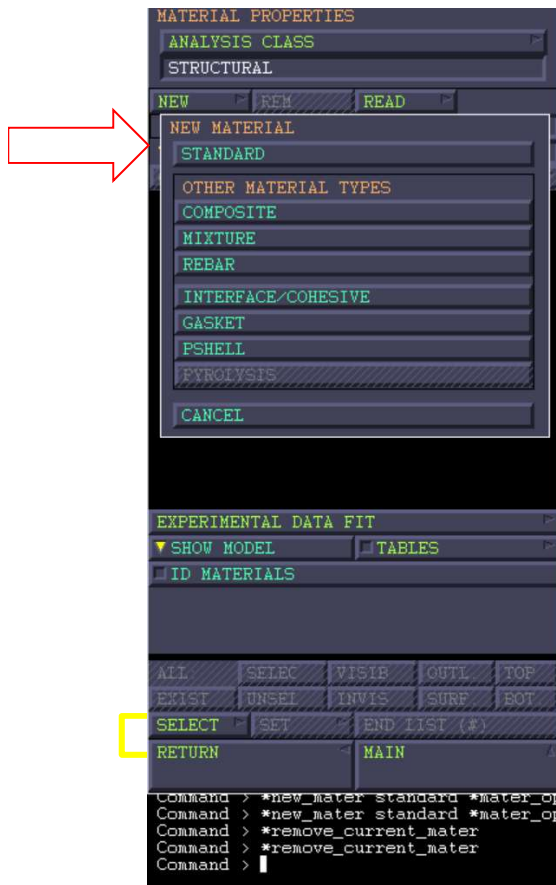
Thin-walled profile in torsion UPDATE

Mesh convergence

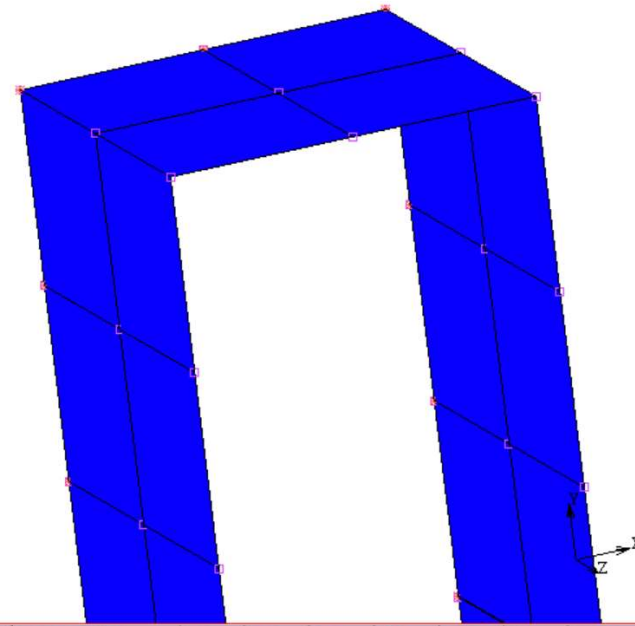
References

Thin-walled profile in torsion

MATERIAL PROPERTIES



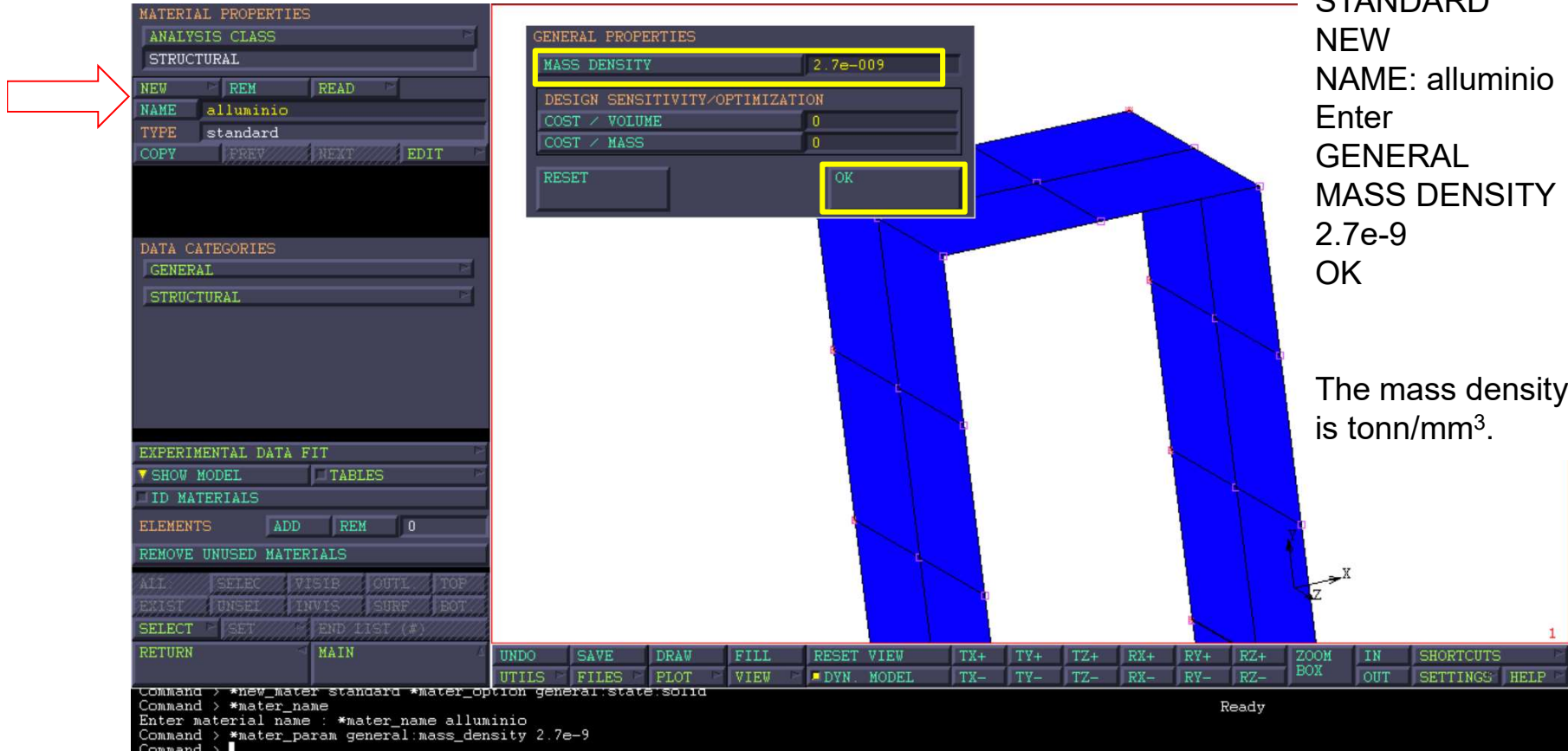
MATERIAL
STANDARD



MeX Software

Thin-walled profile in torsion

MATERIAL PROPERTIES



MATERIAL PROPERTIES

ANALYSIS CLASS
STRUCTURAL

NEW REM READ

NAME alluminio

TYPE standard

COPY PREV NEXT EDIT

DATA CATEGORIES
GENERAL
STRUCTURAL

EXPERIMENTAL DATA FIT
SHOW MODEL TABLES

ID MATERIALS

ELEMENTS ADD REM 0

REMOVE UNUSED MATERIALS

ALL SELEC VISIB OUTL TOP

EXIST UNSEL INVIS SURF BOT

SELECT SET END LIST (#)

RETURN MAIN

GENERAL PROPERTIES

MASS DENSITY 2.7e-009

DESIGN SENSITIVITY/OPTIMIZATION

COST / VOLUME 0

COST / MASS 0

RESET OK

MATERIAL STANDARD
NEW
NAME: alluminio
Enter
GENERAL
MASS DENSITY
2.7e-9
OK

The mass density unit is tonn/mm^3 .

Command > *new_mater standard *mater_option general/state/solid
Command > *mater_name
Enter material name : *mater_name alluminio
Command > *mater_param general:mass_density 2.7e-9
Command >

Ready

Thin-walled profile in torsion

MATERIAL PROPERTIES

STRUCTURAL
YOUNG'S MODULUS
70000
POISSON'S RATIO
0.3
OK

MATERIAL PROPERTIES

ANALYSIS CLASS
STRUCTURAL

NEW REM READ

NAME alluminio

TYPE standard

COPY PREV NEXT EDIT

DATA CATEGORIES

GENERAL

STRUCTURAL

EXPERIMENTAL DATA FIT

SHOW MODEL TABLES

ID MATERIALS

ELEMENTS ADD REM 0

REMOVE UNUSED MATERIALS

ALL SELEC VISIB OUTL TOP

EXIST UNSEL INVIS SURF BOT

SELECT SET END LIST (*)

RETURN MAIN

Command > *new mater standard *mater_option general.state.solid
Command > *mater_name
Enter material name : *mater_name alluminio
Command > *mater_param general.mass_density 2.7e-9
Command >

STRUCTURAL PROPERTIES

TYPE ELASTIC-PLASTIC ISOTROPIC

YOUNG'S MODULUS 0 TABLE

POISSON'S RATIO 0 TABLE

VISCOELASTICITY VISCOPLASTICITY PLASTICITY CREEP

DAMAGE EFFECTS THERMAL EXPANSION CURE SHRINKAGE

DAMPING FORMING LIMIT GRAIN SIZE

RESET OK

UNDO SAVE DRAW FILL RESET VIEW TX+ TY+ TZ+ RX+ RY+ RZ+ ZOOM IN SHORTCUTS
UTILS FILES PLOT VIEW DYN. MODEL TX- TY- TZ- RX- RY- RZ- BOX OUT SETTINGS HELP

Ready

Thin-walled profile in torsion

MATERIAL PROPERTIES

ELEMENTS
ADD
ALL: EXIST
 ID MATERIALS

MATERIAL PROPERTIES

ANALYSIS CLASS
STRUCTURAL

NEW REM READ

NAME alluminio

TYPE standard

COPY PREV NEXT EDIT

DATA CATEGORIES
GENERAL
STRUCTURAL

EXPERIMENTAL DATA FIT
SHOW MODEL TABLES

ID MATERIALS

ELEMENTS ADD REM 32

REMOVE UNUSED MATERIALS

ALL: SELEC VISIB OUTL TOP
EXIST UNSEL INVIS SURE BOT

SELECT SET END LIST (#)

RETURN MAIN

Command > *mater_param structural;poissons_ratio 0.3
Command > *add_mater_elements
Enter add material element list : all_existing
Enter add material element list : *identify_materials *regen
Enter add material element list : .

UNDO SAVE DRAW FILL RESET VIEW TX+ TY+ TZ+ RX+ RY+ RZ+ ZOOM BOX IN SHORTCUTS
UTILS FILES PLOT VIEW DYN. MODEL TX- TY- TZ- RX- RY- RZ- OUT SETTINGS HELP

Ready

Thin-walled profile in torsion

MATERIAL PROPERTIES

The screenshot displays the ANSYS Material Properties dialog box. The left sidebar shows the 'STRUCTURAL' category selected. The main panel shows the 'STRUCTURAL PROPERTIES' for 'ELASTIC-PLASTIC ORTHOTROPIC' material. The material constants are set to 0 for all parameters. A red arrow points to the 'STRUCTURAL' category in the left sidebar.

STRUCTURAL
TYPE: ELASTIC-PLASTIC ORTHOTROPIC
9 MATERIAL CONSTANTS

A local orientation system must be defined!!!!

Thin-walled profile in torsion

MATERIAL PROPERTIES

PLOT
ELEMENTS: SETTING
DRAW:
 WIREFRAME
REGEN

ELEMENT PLOT SETTINGS

DRAW

■ ELEMENTS

◆ SOLID ◆ WIREFRAME

SIZE ▲ 100 % ◆ 90 % ◆ 80 %
 ◆ 70 % ◆ 60 % ◆ 50 %

▢ LABELS

■ FACES

◆ FULL ◆ SURFACE

▢ LABELS ▢ ATTACH INFO

■ EDGES

◆ FULL ◆ SURFACE ◆ OUTLINE

▢ LABELS ▢ ATTACH INFO

FIND SOLID OUTLINE

OUTLINE ANGLE 60

RELATED PLOT SETTINGS

BEAM SHELL

THICKNESS DIRECTION

RESET DRAW REDRAW REGEN

RETURN MAIN

UNDO SAVE DRAW FILL RESET VIEW TX+ TY+ TZ+ RX+ RY+ RZ+ ZOOM IN SHORTCUTS
UTILS FILES PLOT VIEW ■ DYN. MODEL TX- TY- TZ- RX- RY- RZ- BOX OUT SETTINGS HELP

Command > *mater_option structural.type:elast_plast_ortho
Command > *mater_param general:mass_density 2.7e-9
Command > *elements_wireframe
Command > *regenerate
Command > *regenerate
Command > |

Ready

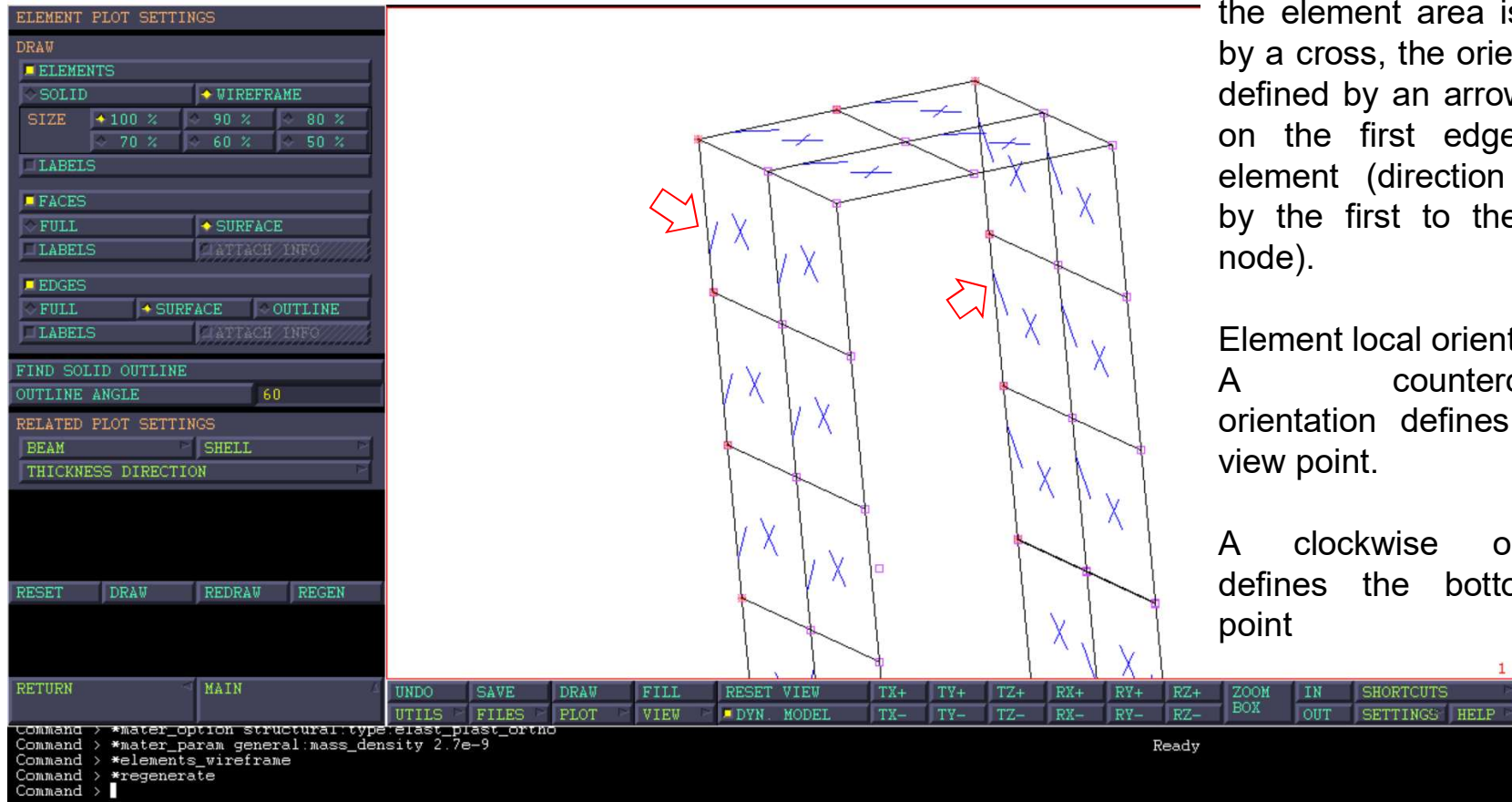
Thin-walled profile in torsion

MATERIAL PROPERTIES

By wireframe visualization, the element area is defined by a cross, the orientation is defined by an arrow located on the first edge of the element (direction defined by the first to the second node).

Element local orientation:
A counterclockwise orientation defines the top view point.

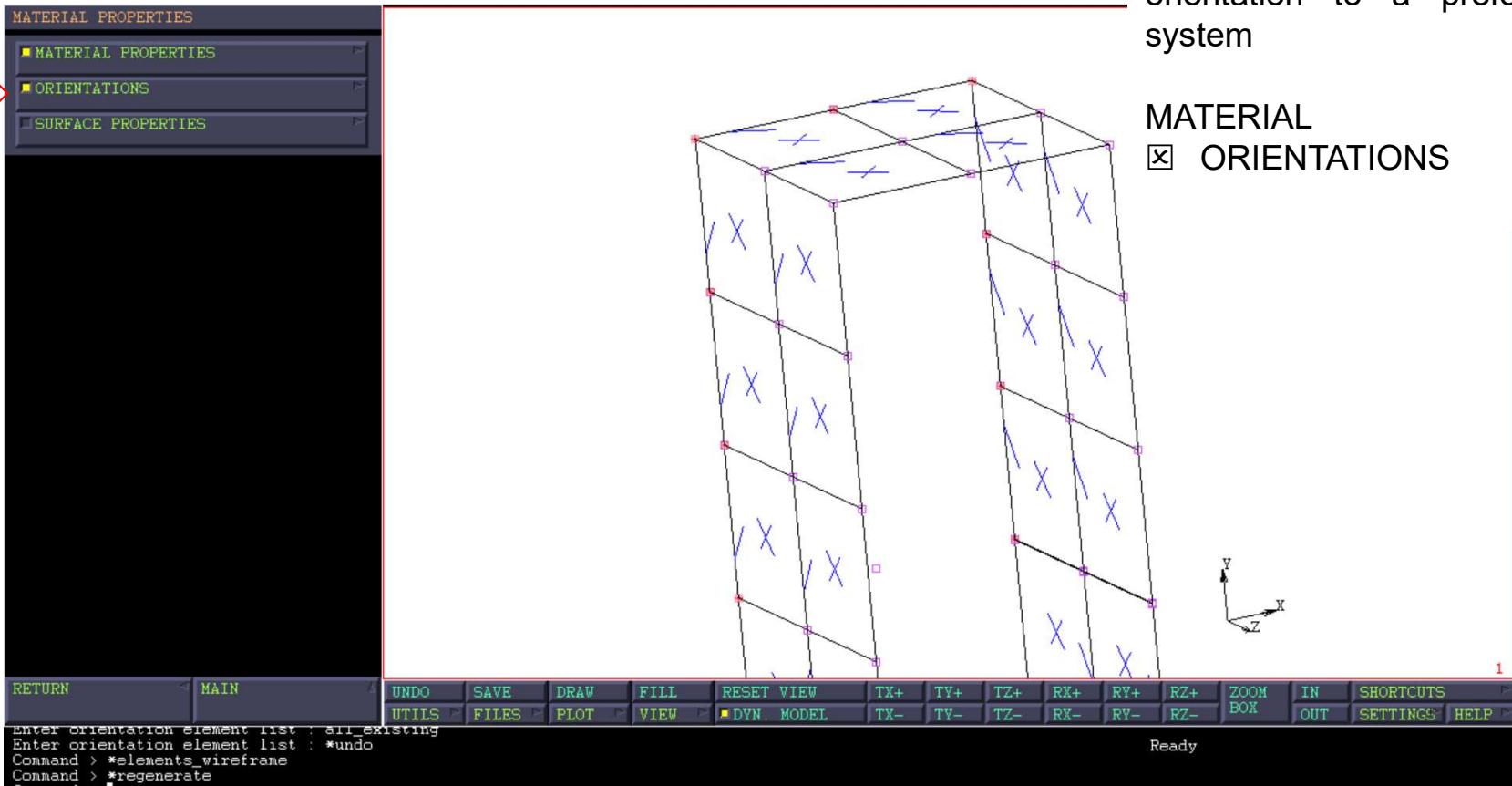
A clockwise orientation defines the bottom view point



Thin-walled profile in torsion

MATERIAL PROPERTIES: orientation

To align the elements orientation to a preferred system



MATERIAL PROPERTIES

- MATERIAL PROPERTIES
- ORIENTATIONS
- SURFACE PROPERTIES

MATERIAL
 ORIENTATIONS

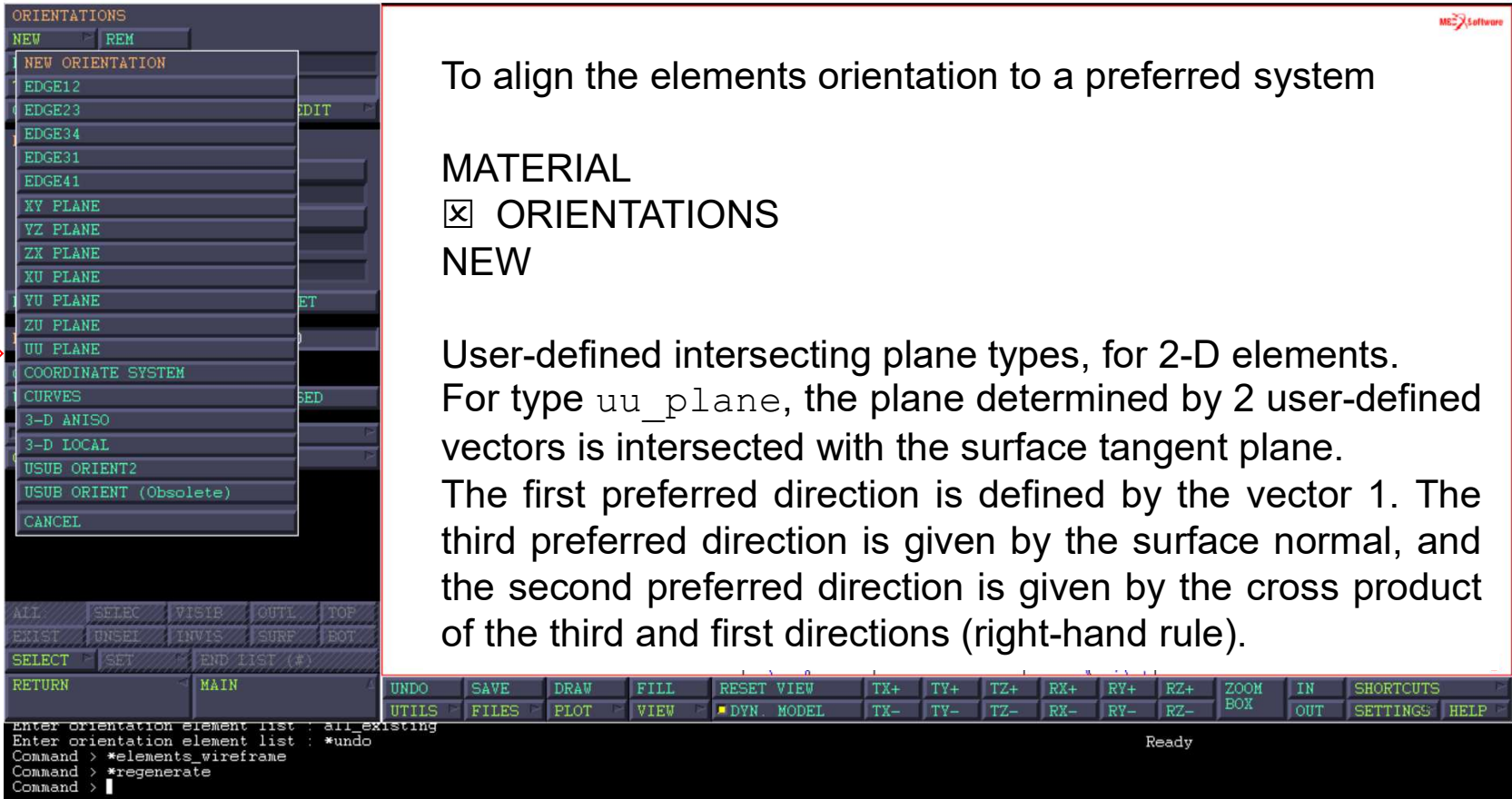
RETURN MAIN UNDO SAVE DRAW FILL RESET VIEW TX+ TY+ TZ+ RX+ RY+ RZ+ ZOOM IN SHORTCUTS
UTILS FILES PLOT VIEW ■ DYN. MODEL TX- TY- TZ- RX- RY- RZ- BOX OUT SETTINGS HELP

Enter orientation element list : all existing
Enter orientation element list : *undo
Command > *elements_wireframe
Command > *regenerate

Ready

Thin-walled profile in torsion

MATERIAL PROPERTIES: orientation



To align the elements orientation to a preferred system

MATERIAL
 ORIENTATIONS
NEW

User-defined intersecting plane types, for 2-D elements.
For type `uu_plane`, the plane determined by 2 user-defined vectors is intersected with the surface tangent plane.
The first preferred direction is defined by the vector 1. The third preferred direction is given by the surface normal, and the second preferred direction is given by the cross product of the third and first directions (right-hand rule).

Enter orientation element list : all_existing
Enter orientation element list : *undo
Command > *elements_wireframe
Command > *regenerate
Command > |

Thin-walled profile in torsion

MATERIAL PROPERTIES: orientation

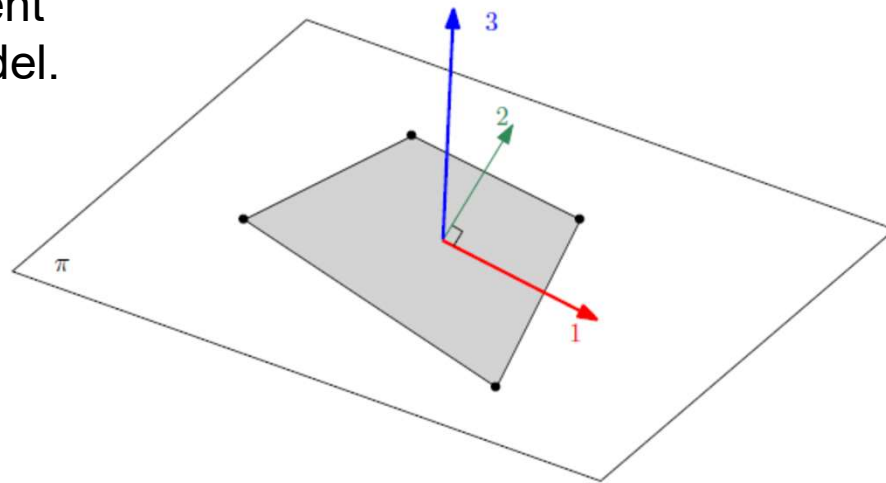
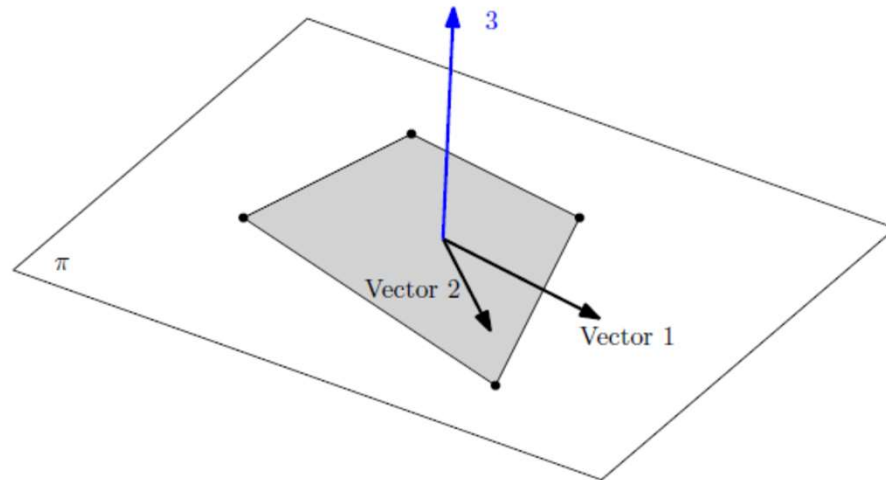
DIRECTION:

1: RED

2: GREEN

3: BLUE

The direction 1 would be coherent with the global axis Z of the model.



Thin-walled profile in torsion

MATERIAL PROPERTIES: orientation

To align the elements orientation to a preferred system

The screenshot displays the 'ORIENTATIONS' menu in a CAD application. The menu is open, showing options for defining new orientations and coordinate systems. Two red arrows point to the 'NEW ORIENTATION' section and the 'UU PLANE' option. The 3D model shows a thin-walled profile with blue 'X' marks indicating the orientation of the elements. A coordinate system (X, Y, Z) is visible in the bottom right corner of the model area. The software interface includes a command line at the bottom with the following text:

```
Enter orientation element list : all_existing
Enter orientation element list : *undo
Command > *elements_wireframe
Command > *regenerate
Command > |
```

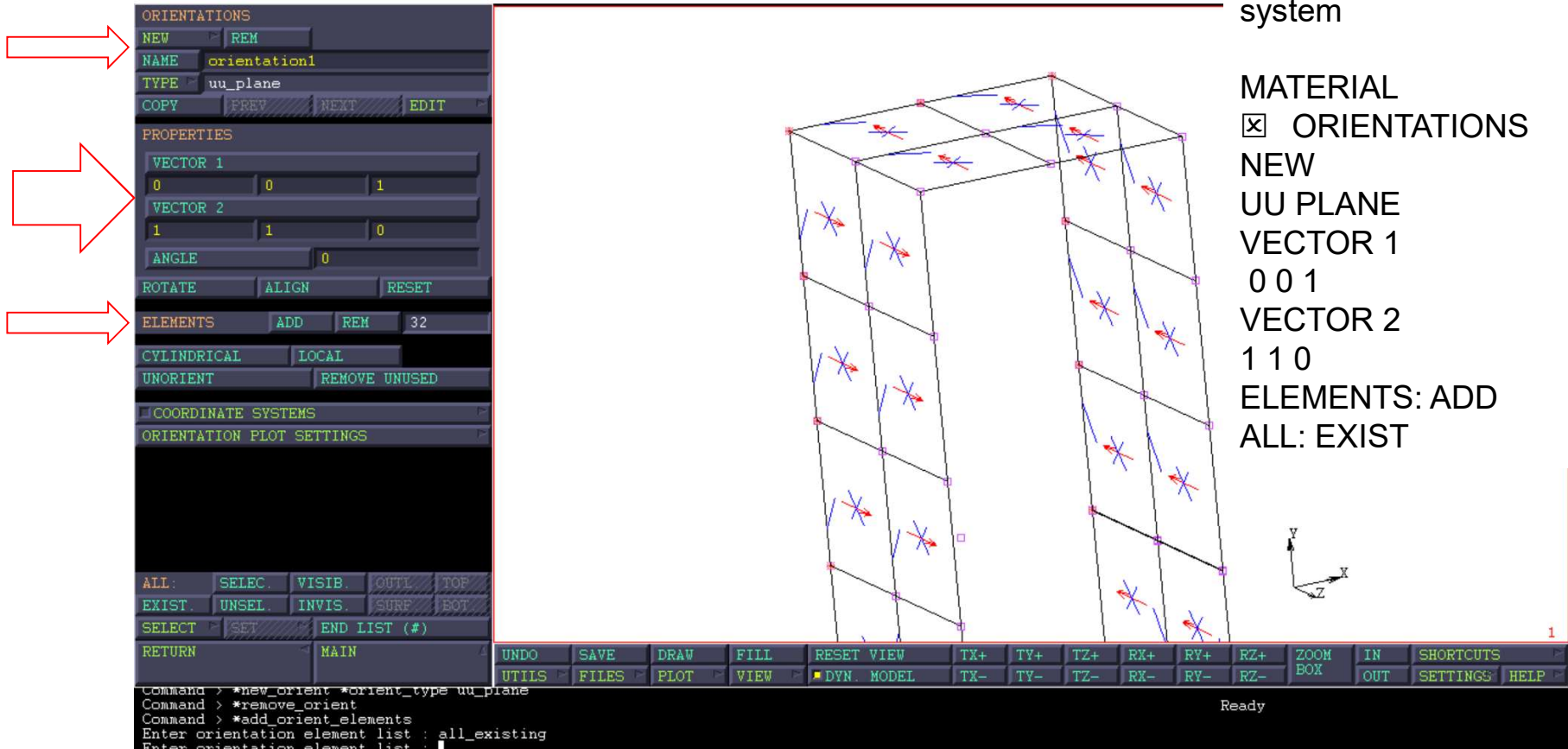
The software interface also includes a toolbar with various functions such as UNDO, SAVE, DRAW, FILL, RESET VIEW, TX+, TY+, TZ+, RX+, RY+, RZ+, ZOOM BOX, IN, SHORTCUTS, UTILS, FILES, PLOT, VIEW, DYN. MODEL, TX-, TY-, TZ-, RX-, RY-, RZ-, OUT, SETTINGS, and HELP. The status bar at the bottom right shows 'Ready'.

MATERIAL
 ORIENTATIONS
NEW
UU PLANE

Thin-walled profile in torsion

MATERIAL PROPERTIES: orientation

To align the elements orientation to a preferred system



The screenshot displays the 'ORIENTATIONS' dialog box on the left, which is used to define the orientation of elements in a finite element model. The dialog box includes the following fields and options:

- ORIENTATIONS**
 - NEW: REM
 - NAME: orientation1
 - TYPE: uu_plane
 - COPY: PREV NEXT EDIT
- PROPERTIES**
 - VECTOR 1: 0 0 1
 - VECTOR 2: 1 1 0
 - ANGLE: 0
 - ROTATE: ALIGN RESET
- ELEMENTS**: ADD REM 32
- COORDINATE SYSTEMS**: LOCAL
- ORIENTATION PLOT SETTINGS**
- ALL:** SELEC. VISIB. OUTL. TOP
- EXIST.** UNSEL. INVIS. SURF. BOT
- SELECT**: SET END LIST (#)
- RETURN**: MAIN

The 3D model on the right shows a thin-walled profile with a grid of elements. Blue arrows indicate the orientation of the elements, which are aligned with the preferred system defined in the dialog box. A small coordinate system (X, Y, Z) is visible in the bottom right corner of the model view.

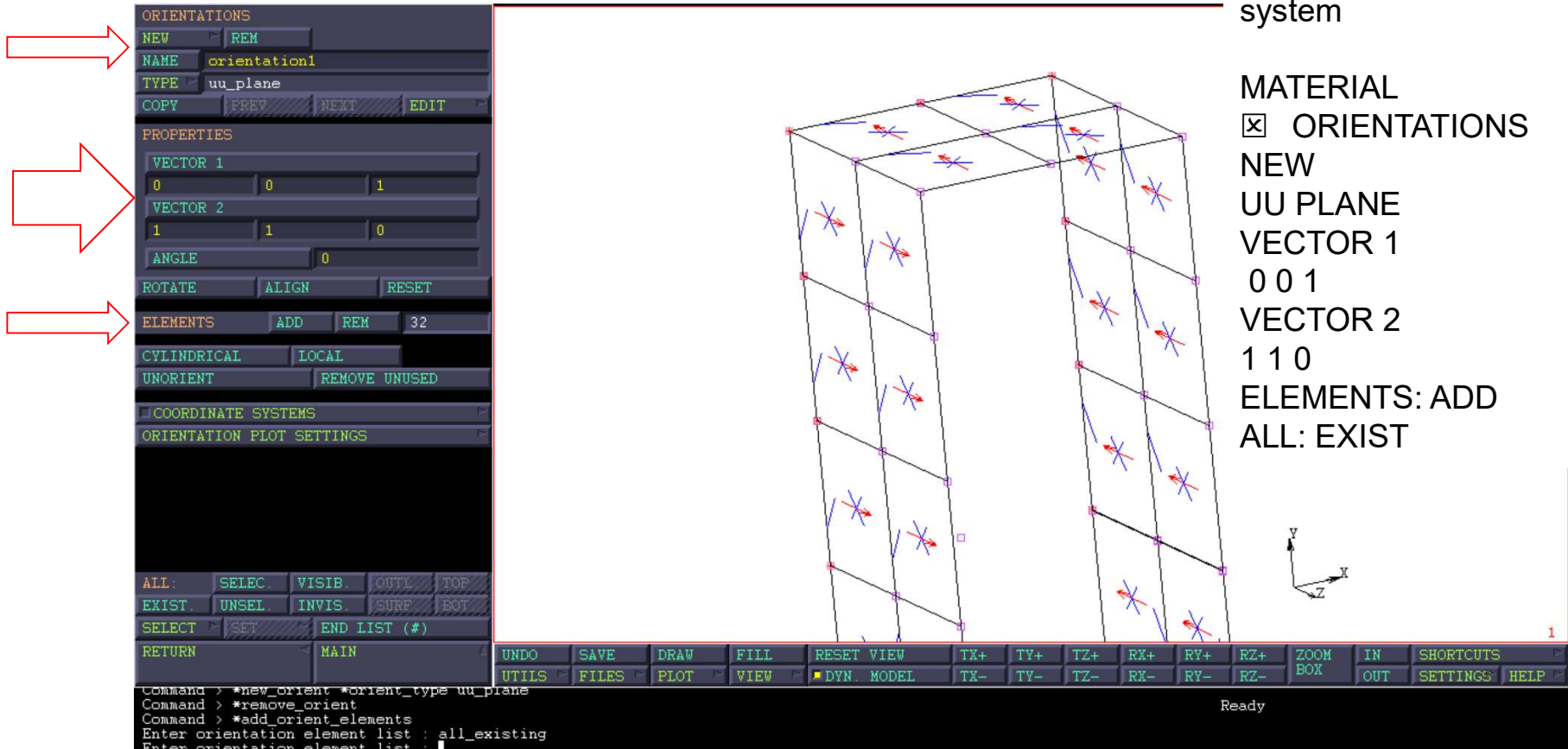
MATERIAL
 ORIENTATIONS
NEW
UU PLANE
VECTOR 1
0 0 1
VECTOR 2
1 1 0
ELEMENTS: ADD
ALL: EXIST

```
Command > *new_orient *orient_type uu_plane
Command > *remove_orient
Command > *add_orient_elements
Enter orientation element list : all_existing
Enter orientation element list :
```

Thin-walled profile in torsion

MATERIAL PROPERTIES: orientation

To align the elements orientation to a preferred system



The screenshot displays the 'ORIENTATIONS' dialog box on the left, which is used to define the orientation of elements in a finite element model. The dialog box includes the following fields and options:

- ORIENTATIONS**
 - NEW: REM
 - NAME: orientation1
 - TYPE: uu_plane
 - COPY: PREV NEXT EDIT
- PROPERTIES**
 - VECTOR 1: 0 0 1
 - VECTOR 2: 1 1 0
 - ANGLE: 0
 - ROTATE: ALIGN RESET
- ELEMENTS**: ADD REM 32
- COORDINATE SYSTEMS**: LOCAL
- ORIENTATION PLOT SETTINGS**
- ALL:** SELEC. VISIB. OUTL. TOP
- EXIST.** UNSEL. INVIS. SURF. BOT
- SELECT:** SET END LIST (#)
- RETURN:** MAIN

The 3D model on the right shows a thin-walled profile with a grid of elements. Blue arrows indicate the orientation of the elements, which are aligned with the preferred system defined in the dialog box. A small coordinate system (X, Y, Z) is visible in the bottom right corner of the model view.

MATERIAL
 ORIENTATIONS
NEW
UU PLANE
VECTOR 1
0 0 1
VECTOR 2
1 1 0
ELEMENTS: ADD
ALL: EXIST

Command > *new_orient *orient_type uu_plane
Command > *remove_orient
Command > *add_orient_elements
Enter orientation element list : all_existing
Enter orientation element list :

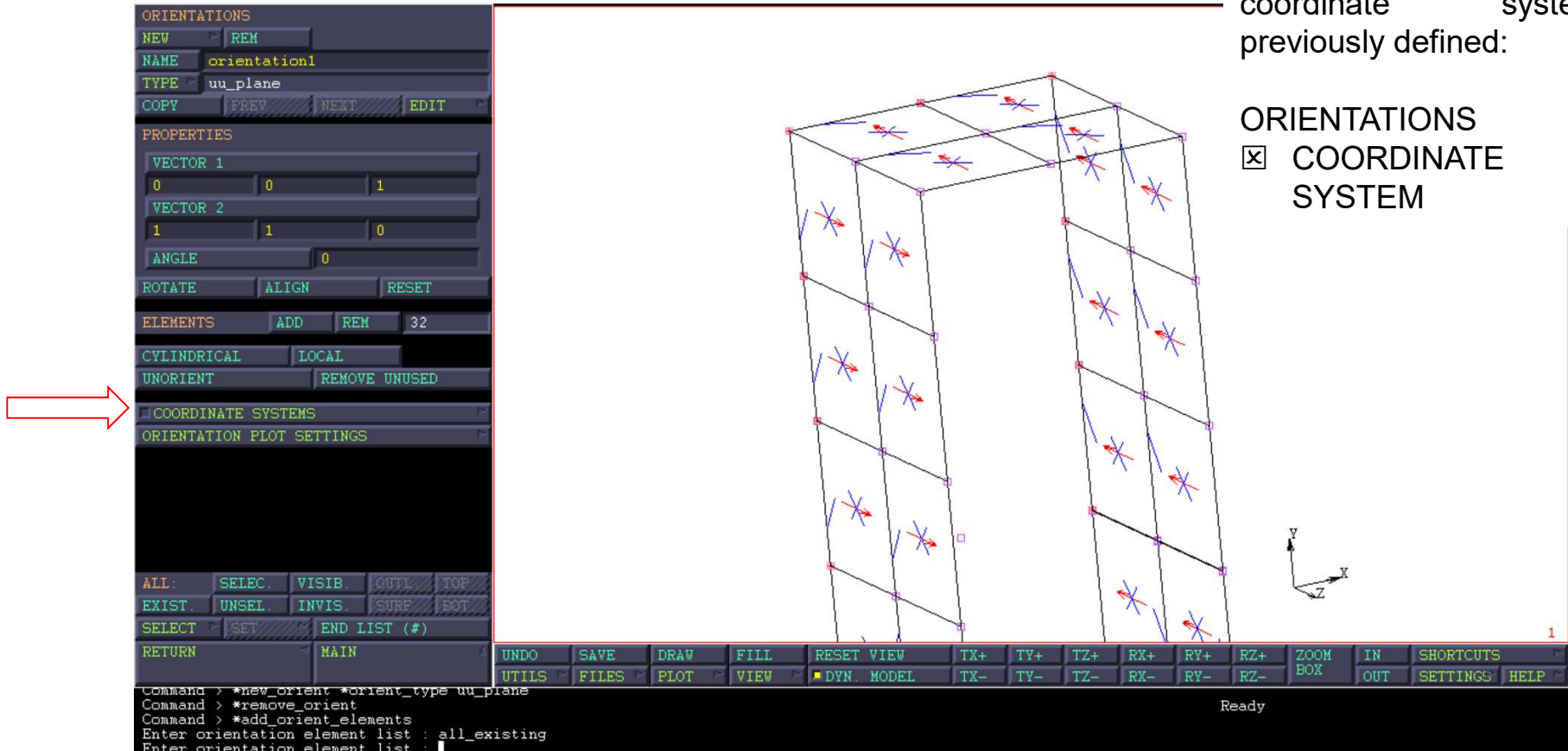
Ready

Thin-walled profile in torsion

MATERIAL PROPERTIES: orientation

To plot the local coordinate system previously defined:

ORIENTATIONS
 COORDINATE SYSTEM



The image shows a screenshot of a finite element software interface. On the left, a panel displays the 'ORIENTATIONS' settings for a local coordinate system named 'orientation1'. The 'TYPE' is set to 'uu_plane'. The 'VECTOR 1' is (0, 0, 1) and 'VECTOR 2' is (1, 1, 0). The 'ANGLE' is 0. The 'ELEMENTS' list contains 32 elements. A red arrow points to the 'COORDINATE SYSTEMS' section. The main window shows a 3D model of a thin-walled profile with local coordinate systems (red and blue axes) plotted on each element. A small 3D coordinate system is visible in the bottom right corner. The bottom status bar shows 'Ready'.

```
Command > *new_orient *orient_type uu_plane
Command > *remove_orient
Command > *add_orient_elements
Enter orientation element list : all_existing
Enter orientation element list :
```


Thin-walled profile in torsion

MATERIAL PROPERTIES: orientation

To plot the local coordinate system previously defined:

The screenshot displays the 'ORIENTATION PLOT SETTINGS' dialog box on the left. Under the 'DRAW' section, the 'ORIENTATIONS' dropdown is set to 'CUSTOM', which is highlighted by a red arrow. Other options include 'FIRST DIRECTION', 'SECOND DIRECTION', 'THIRD DIRECTION', and 'LABELS'. Below this, the 'ARROW PLOT SETTINGS' section is visible. At the bottom of the dialog are buttons for 'RESET', 'DRAW', 'REDRAW', and 'REGEN'. The main window shows a 3D model of a thin-walled profile with a grid of local coordinate systems (X, Y, Z axes) plotted on its surface. A small 3D coordinate system icon is also visible in the bottom right corner of the main window. The bottom status bar shows various tool icons and the text 'Ready'. The command line at the bottom left shows the following commands: 'Enter orientation element list : *set_orient_custom on', 'Command > *regenerate', 'Command > *set_orient_custom off', and 'Command > *set_orient_custom on'.

ORIENTATIONS
 COORDINATE SYSTEM
DRAW:
 CUSTOM

Agenda

Finite Element Modelling

MSC Marc Mentat

Thin-walled profile in torsion

- The problem
- Mesh generation
- Geometric properties
- Material properties
- [Links: RBE2](#)
- Boundary conditions
- Jobs
- Results

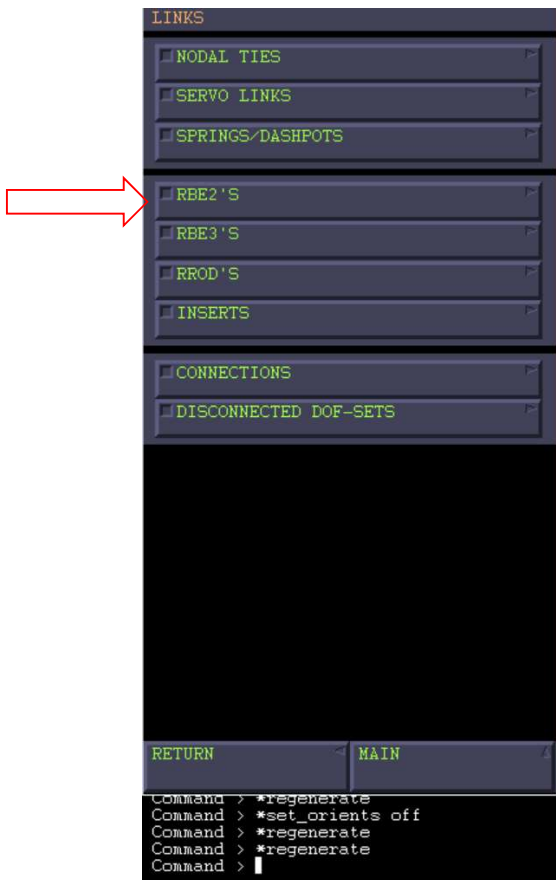
Thin-walled profile in torsion UPDATE

Mesh convergence

References

Thin-walled profile in torsion

LINKS: RBE2



LINKS = KINEMATIC RELATIONS BETWEEN NODES
The RBE2 is a nodal tie, it is a rigid link type of tie.
A number of **tied nodes** are rigidly connected to a **retained node (ONLY ONE)**.
The retained node can have loads and boundary conditions applied to it.
The tying can be done for separate degrees of freedom or for all degrees of freedom.
The RBE2 supports large deformations and rotations.

In the present model, the prescribed rotation will be applied to the retained node of the RBE2, as described in the following.

Command > *regenerate
Command > *set_orients off
Command > *regenerate
Command > *regenerate
Command > |

Thin-walled profile in torsion

LINKS: RBE2

To create a RBE2:

RBE2
NEW
RETAINED
NODE
104
TIED NODES: ADD

Enter remove_rbe2_tied_nodes list : *add_rbe2_tied_nodes
Enter add_rbe2_tied_nodes list : *clear_rbe2_tied_dof 1
Command > *clear_rbe2_tied_dof 2
Command > *clear_rbe2_tied_dof 6
Command > |

Thin-walled profile in torsion

LINKS: RBE2

To create a RBE2:

RBE2
NEW
RETAINED
NODE
104
TIED NODES: ADD

Enter remove_rbe2_tied_nodes list : *add_rbe2_tied_nodes
Enter add_rbe2_tied_nodes list : *clear_rbe2_tied_dof 1
Command > *clear_rbe2_tied_dof 2
Command > *clear_rbe2_tied_dof 6
Command > |

Thin-walled profile in torsion

LINKS: RBE2

To create a RBE2:

RBE2'S

NEW REM
NAME rbe2_1
COPY PREV NEXT EDIT PICK

RETAINED (REFERENCE)
NODE 102

TIED NODES
ADD REM 0
DOF 1 2 3 4 5 6

DEACTIVATED ANALYSIS PASSES
REM RBE2'S REM ALL RBE2'S

CURRENTLY DEFINED SETS

retained_node_z40	node	1
tied_nodes_z40	node	17

OK

ALL: SELEC VISIB OUTL TOP
EXIST UNSET INVIS SURF BOT
SELECT SET END LIST (#)
RETURN MAIN

UNDO SAVE DRAW FILL RESET VIEW TX+ TY+ TZ+ RX+ RV+ RZ+ ZOOM IN SHORTCUTS
UTILS FILES PLOT VIEW DYN. MODEL TX- TY- TZ- RX- RV- RZ- ZOOM BOX OUT SETTINGS HELP

Enter add rbe2 tied nodes list : *clear_rbe2_tied_dof 1
Command > *clear_rbe2_tied_dof 2
Command > *clear_rbe2_tied_dof 6
Command > *add_rbe2_tied_nodes

Ready

1

RBE2
NEW
RETAINED
NODE
104
TIED NODES: ADD
SET
tied_nodes_z40

Thin-walled profile in torsion

LINKS: RBE2

To create a RBE2:

RBE2
NEW
RETAINED
NODE
104
TIED NODES: ADD
SET
tied_nodes_z40
DOF:
 1
 2
 6

skew_symm_xy_nodes
Enter show node : *dynamic_model_on
Enter show node : *undo
Command > *add_apply_nodes
Enter add apply node list :

Agenda

Finite Element Modelling

MSC Marc Mentat

Thin-walled profile in torsion

- The problem
- Mesh generation
- Geometric properties
- Material properties
- Links: RBE2
- **Boundary conditions**
- Jobs
- Results

Thin-walled profile in torsion UPDATE

Mesh convergence

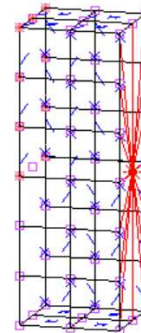
References

Thin-walled profile in torsion

BOUNDARY CONDITIONS

BCs: xy-skew symmetry
definition at coordinate $z = 0$

BOUNDARY CONDITIONS
NEW
STRUCTURAL
FIXED DISPLACEMENT

A screenshot of the ANSYS software interface. The 'BOUNDARY CONDITIONS' menu is open, showing a list of options. The 'NEW' button is highlighted, and the 'NEW STRUCTURAL BC' sub-menu is also open, showing a list of boundary condition types. The 'FIXED DISPLACEMENT' option is selected. The software interface includes a command window at the bottom with the following text:

```
Enter remove rbe2 tied nodes list : retained_node_240
Enter remove rbe2 tied nodes list : *save model
Model file c:\users\manto\documents\201819_esami\paom\20190328_paom_lab\torsione_rev00.mud opened for writing.
Model saved to c:\users\manto\documents\201819_esami\paom\20190328_paom_lab\torsione_rev00.
Enter remove rbe2 tied nodes list :
```

Thin-walled profile in torsion

BOUNDARY CONDITIONS

BOUNDARY CONDITIONS

ANALYSIS CLASS
STRUCTURAL

NEW REM

NAME skew_symm_xy

TYPE fixed_displacement

COPY PREV NEXT EDIT

PROPERTIES

PLOT SETTINGS

DRAW BOUNDARY CONDS ON MESH

ID BOUNDARY CONDS

ARROW PLOT SETTINGS

MERGE DUPLICATE BOUND CONDS

REMOVE ALL BOUND CONDS

TABLES TRANSFORMATIONS

NODES ADD REM 0

POINTS ADD REM 0

CURVES ADD REM 0

SURFACES ADD REM 0

ALL: SELEC. VISIB. OUTL. TOP

EXIST. UNSEL. INVIS. SURF. BOT

SELECT SET END LIST (#)

RETURN MAIN UNDO SAVE DRAW

UTILS FILES PLOT

Command > *clear_apply_dof rx
Command > *clear_apply_dof ry
Command > *remove_apply_nodes
Enter remove apply node list : all_existing
Enter remove apply node list :

FIXED DISPLACEMENT

METHOD ENTERED VALUES

REFERENCE POSITION POSITION AT ACTIVATION OF BC

TIME DEPENDENCE TABLES

DISPLACEMENT X 0 TABLE

DISPLACEMENT Y 0 TABLE

DISPLACEMENT Z

ROTATION X

ROTATION Y

ROTATION Z 0 TABLE

CLEAR OK

BCs:xy-skew-symmetry definition at coordinate z = 0

BOUNDARY CONDITIONS
NEW
STRUCTURAL
NAME: skew_symm_xy
TYPE: FIXED DISPLACEMENT
 DISPLACEMENT X
 DISPLACEMENT Y
 ROTATION Z
OK

Thin-walled profile in torsion

BOUNDARY CONDITIONS

BOUNDARY CONDITIONS

ANALYSIS CLASS
STRUCTURAL

NEW REM

NAME skew_symm_xy

TYPE fixed_displacement

COPY PREV NEXT EDIT

PROPERTIES

PLOT SETTINGS

DRAW BOUNDARY CONDS ON MESH

ID BOUNDARY CONDS

ARROW PLOT SETTINGS

MERGE DUPLICATE BOUND CONDS

REMOVE ALL BOUND CONDS

TABLES TRANSFORMATIONS

NODES ADD REM 0

POINTS ADD REM 0

CURVES ADD REM 0

SURFACES ADD REM 0

ALL: SELEC VISIB OUTL TOP

EXIST UNSEL UNVIS SURF BOT

SELECT SET END LIST (#)

RETURN MAIN

skew_symm_xy_nodes

Enter show node : *dynamic_model_on

Enter show node : *undo

Command > *add_apply_nodes

Enter add apply node list :

CURRENTLY DEFINED SETS

rbe2_1_nodes	node	17
retained_node_z40	node	1
skew_symm_nodes_xy	node	17
tied_nodes_z40	node	17

OK

BCs:xy-skew-symmetry
definition at coordinate z = 0

NODES: ADD
SET
skew_symm_nodes_xy
OK

Ready

2 nuove notifiche

Thin-walled profile in torsion

BOUNDARY CONDITIONS

BOUNDARY CONDITIONS

ANALYSIS CLASS
STRUCTURAL

NEW REM
NAME skew_symm_xy
TYPE fixed_displacement
COPY PREV NEXT EDIT

PROPERTIES

PLOT SETTINGS
DRAW BOUNDARY CONDS ON MESH
ID BOUNDARY CONDS
ARROW PLOT SETTINGS

MERGE DUPLICATE BOUND CONDS
REMOVE ALL BOUND CONDS

TABLES TRANSFORMATIONS

NODES ADD REM 17
POINTS ADD REM 0
CURVES ADD REM 0
SURFACES ADD REM 0

ALL SELEC VISIB OUTL TOP
EXIST UNSEL INVIS SURF BOT
SELECT SET END LIST (#)
RETURN MAIN

UNDO SAVE DRAW FILL RESET VIEW TX+ TY+ TZ+ RX+ RY+ RZ+ ZOOM IN SHORTCUTS
UTILS FILES PLOT VIEW DYN. MODEL TX- TY- TZ- RX- RY- RZ- BOX OUT SETTINGS HELP

Enter value for 'x' : *apply_dof y *apply_dof_value y
Enter value for 'y' : *apply_dof rz *apply_dof_value rz
Enter value for 'rz' : *add_apply_nodes
Enter add apply node list : skew_symm_nodes_xy
Enter add apply node list :

Ready

Thin-walled profile in torsion

SKEW-SYMM and RBE2 dof

BOUNDARY CONDITIONS

NAME: skew_symm_xy

TYPE: FIXED DISPLACEMENT

- DISPLACEMENT X
- DISPLACEMENT Y
- ROTATION Z

RBE2

NEW

RETAINED

NODE

104

TIED NODES: ADD

SET

tied_nodes_z40

DOF:

- 1
- 2
- 6



→ The rigid element dofs constraints are equal to the skew-symm, therefore, the retained node of RBE2 might be thought as a skew-symm BCs, free referring to Z displacement, and free to the rotations associated to Z.

Thin-walled profile in torsion

BCs: rotation Z prescribed

The imposed rotation Z must be 0.001 rad/mm, this rotation relies on the length of the profile.

BOUNDARY CONDITIONS
NEW
STRUCTURAL
FIXED DISPLACEMENT

The screenshot displays the 'BOUNDARY CONDITIONS' menu with 'NEW STRUCTURAL BC' selected. The 'NEW STRUCTURAL BC' dialog box is open, showing 'FIXED DISPLACEMENT' as the chosen option. The command window at the bottom shows the following commands:

```
Enter show node : *undo  
Command > *add_apply_nodes  
Enter add apply node list : *add_apply_nodes  
Enter add apply node list : skew_symm_nodes_xy  
Enter add apply node list : |
```

Thin-walled profile in torsion

BCs: rotation Z prescribed

The imposed rotation Z must be 0.001 rad/mm, this rotation relies on the length of the profile.

BOUNDARY CONDITIONS
NEW
STRUCTURAL
FIXED DISPLACEMENT
 TABLES

The screenshot displays a software interface for defining boundary conditions. On the left, a sidebar contains a 'BOUNDARY CONDITIONS' panel with the following settings: ANALYSIS CLASS: STRUCTURAL; NAME: rotation_z; TYPE: fixed_displacement. Below this are 'PLOT SETTINGS' and 'TABLES' options. A red arrow points to the 'TABLES' section, which includes a table for defining nodes, points, curves, and surfaces. The main window shows a 3D wireframe model of a thin-walled profile with a mesh of blue 'x' marks and red lines indicating the boundary conditions. A small 3D coordinate system (X, Y, Z) is visible in the bottom right corner of the main window. The bottom status bar shows 'Ready' and various utility buttons like UNDO, SAVE, DRAW, FILL, RESET VIEW, etc.

TABLES	TRANSFORMATIONS
NODES	ADD REM 0
POINTS	ADD REM 0
CURVES	ADD REM 0
SURFACES	ADD REM 0

```
Command > *remove_current_table
Command > *clear_current_apply
Command > *remove_current_table
Command > *apply_type fixed_displacement @popdown(apply_type_pm)
Command >
```

Thin-walled profile in torsion

BCs: rotation Z prescribed

The imposed rotation Z must be 0.001 rad/mm, this rotation relies on the length of the profile.

BOUNDARY CONDITIONS

NEW

STRUCTURAL

FIXED DISPLACEMENT

TABLES

1 INDEPENDENT VARIABLES

The screenshot shows a software interface with a 'TABLES' dialog box on the left and a command line at the bottom. A red arrow points to the 'NEW TABLE' button in the dialog. The dialog box has a 'NEW TABLE' section with options: '1 INDEPENDENT VARIABLE', '2 INDEPENDENT VARIABLES', '3 INDEPENDENT VARIABLES', '4 INDEPENDENT VARIABLES', 'EXPERIMENTAL DATA FIT', '1 INDEP. & 2 DEP. VARIABLES', and 'CANCEL'. Below the dialog are various plot and table controls. The command line at the bottom shows the following commands: 'Enter value for rx : *apply_dof_value rz 0.0005', 'Command > *apply_dof_table rz', 'Enter boundary condition D.O.F. table name : *new_md_table 1 1', 'Command > *remove_current_table', and 'Command > |'. The status bar at the bottom right says 'Ready'.

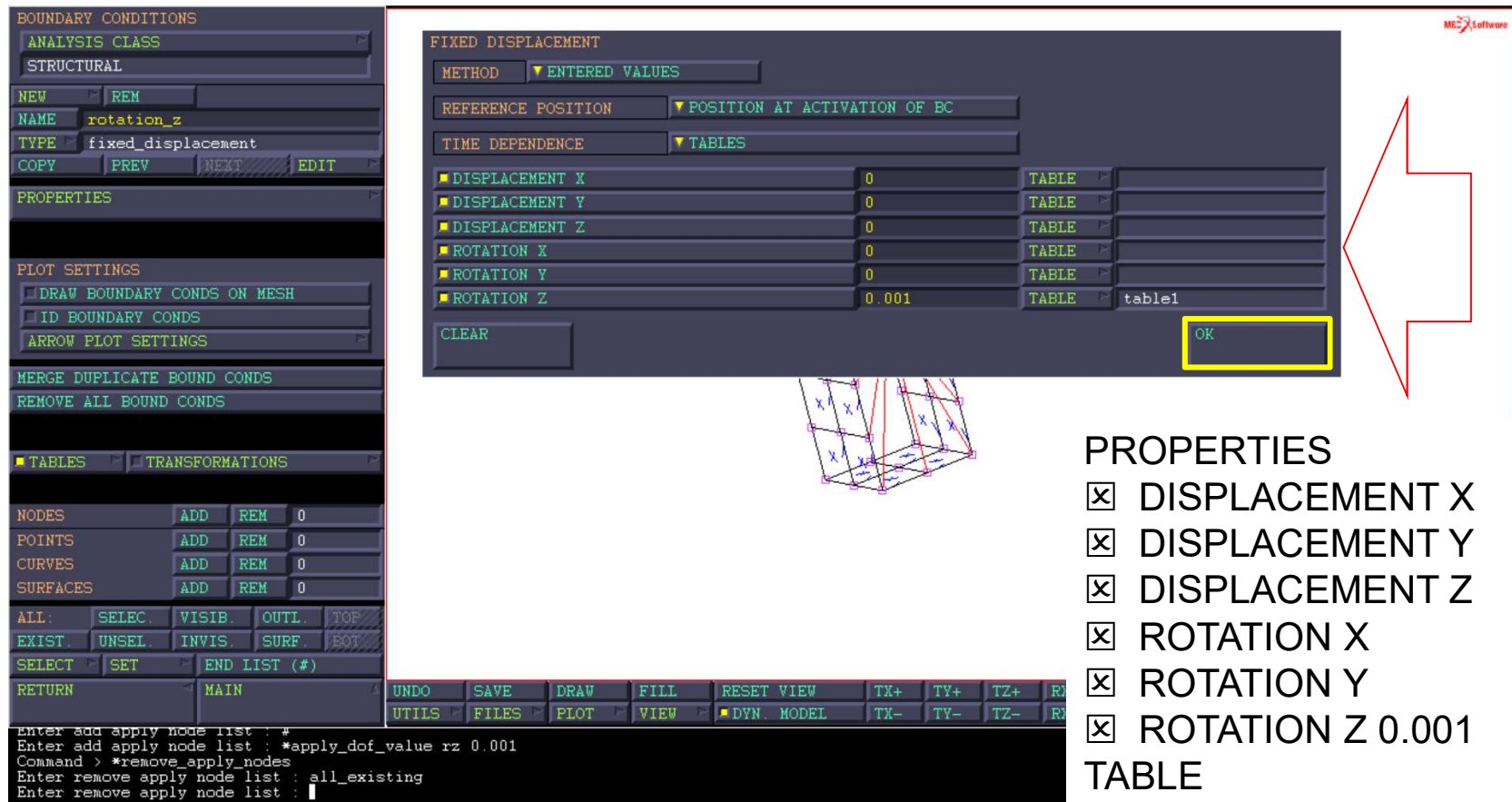
Thin-walled profile in torsion

BCs: rotation Z prescribed

The screenshot displays a software interface for configuring a table and viewing a graph. On the left, the 'TABLES' panel shows the configuration for 'table1'. The 'VARIABLES' section is set to 'INDEPENDENT VARIABLE V1' with 'TYPE' as 'z0_coordinate', 'MIN' as 0, 'MAX' as 1, and 'STEPS' as 10. The 'DATA POINTS' section is set to 'FORMULA'. The 'FUNCTION VALUE F' section is also configured. The graph on the right shows a linear plot of 'F' versus 'V1' with 11 data points labeled 1 through 11. A text box on the graph reads 'NEW TYPE: z0_coordinate [x] FORMULA v1'. The bottom status bar shows 'Ready'.

Thin-walled profile in torsion

BCs: rotation Z prescribed



BOUNDARY CONDITIONS

ANALYSIS CLASS
STRUCTURAL

NEW REM
NAME rotation_z
TYPE fixed_displacement
COPY PREV NEXT EDIT

PROPERTIES

PLOT SETTINGS
DRAW BOUNDARY CONDS ON MESH
ID BOUNDARY CONDS
ARROW PLOT SETTINGS

MERGE DUPLICATE BOUND CONDS
REMOVE ALL BOUND CONDS

TABLES TRANSFORMATIONS

NODES ADD REM 0
POINTS ADD REM 0
CURVES ADD REM 0
SURFACES ADD REM 0

ALL: SELEC VISIB OUTL TOP
EXIST UNSEL INVIS SURF BOT
SELECT SET END LIST (#)
RETURN MAIN

FIXED DISPLACEMENT

METHOD ENTERED VALUES

REFERENCE POSITION POSITION AT ACTIVATION OF BC

TIME DEPENDENCE TABLES

DISPLACEMENT X	0	TABLE
DISPLACEMENT Y	0	TABLE
DISPLACEMENT Z	0	TABLE
ROTATION X	0	TABLE
ROTATION Y	0	TABLE
ROTATION Z	0.001	TABLE table1

CLEAR OK

PROPERTY LIST

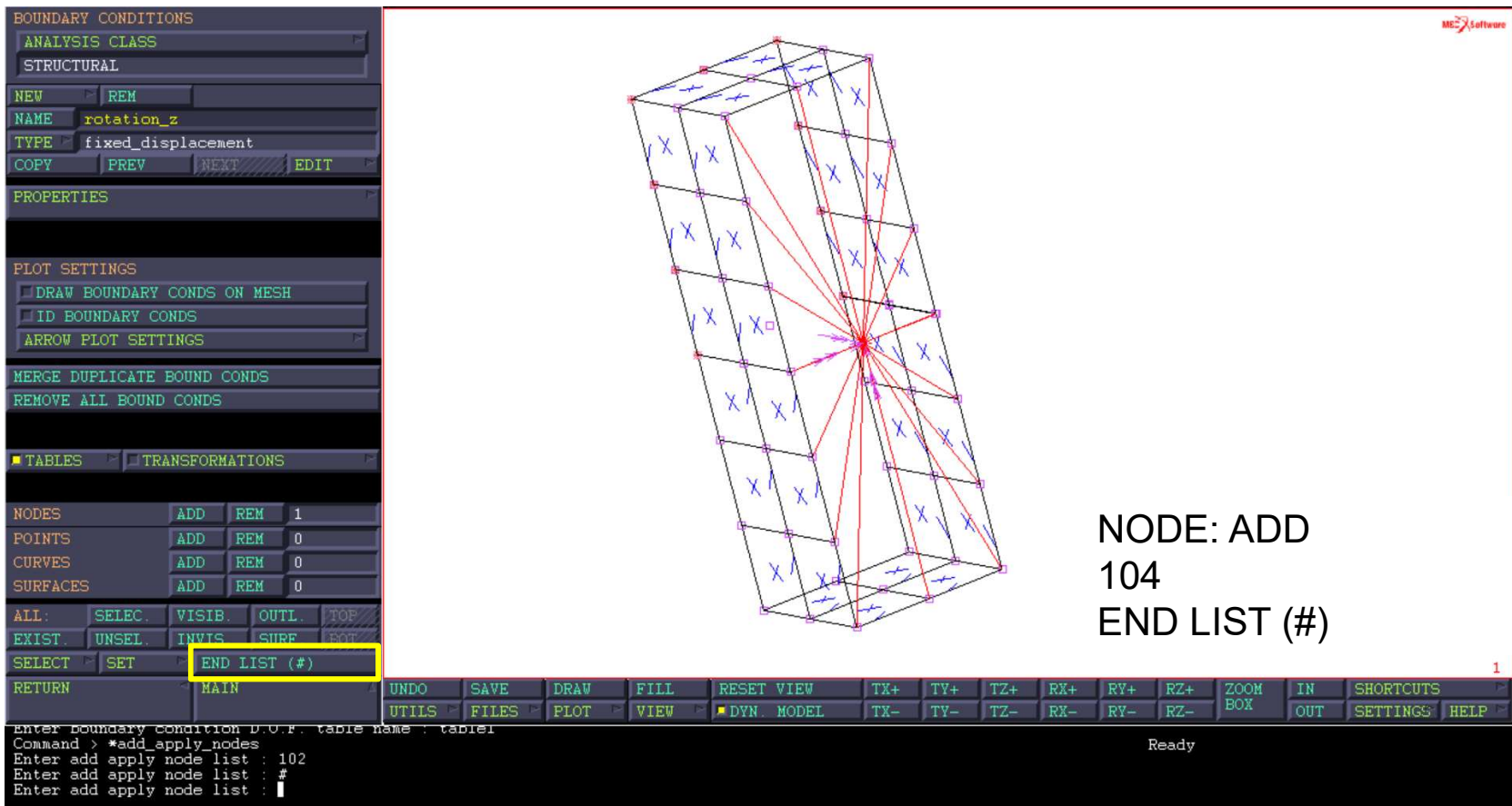
- DISPLACEMENT X
- DISPLACEMENT Y
- DISPLACEMENT Z
- ROTATION X
- ROTATION Y
- ROTATION Z 0.001

TABLE
table1
OK

```
Enter add apply node list : #  
Enter add apply node list : *apply_dof_value rz 0.001  
Command > *remove_apply_nodes  
Enter remove apply node list : all_existing  
Enter remove apply node list :
```

Thin-walled profile in torsion

BCs: rotation Z prescribed



The screenshot displays the 'BOUNDARY CONDITIONS' panel in a finite element software. The 'ANALYSIS CLASS' is set to 'STRUCTURAL'. The 'NAME' is 'rotation_z' and the 'TYPE' is 'fixed_displacement'. A table under the 'TABLES' tab shows the following data:

	ADD	REM	
NODES			1
POINTS			0
CURVES			0
SURFACES			0

The 3D model shows a thin-walled profile with a red dot at the center of the front face, representing the prescribed rotation. The command window at the bottom shows the following commands:

```
Enter boundary condition D.O.F. table name : table1  
Command > *add_apply_nodes  
Enter add apply node list : 102  
Enter add apply node list : #  
Enter add apply node list : |
```

Below the 3D model, the text reads:

NODE: ADD
104
END LIST (#)

Agenda

Finite Element Modelling

MSC Marc Mentat

Thin-walled profile in torsion

- The problem
- Mesh generation
- Geometric properties
- Material properties
- Links: RBE2
- Boundary conditions
- **Jobs**
- Results

Thin-walled profile in torsion UPDATE

Mesh convergence

References

Thin-walled profile in torsion

JOBS

The screenshot shows the ANSYS software interface. On the left, the 'MAIN MENU' is open, with the 'JOBS' option highlighted under the 'ANALYSIS' section. A red arrow points to the 'JOBS' option. The main window displays a 3D model of a thin-walled profile (a rectangular tube) with a mesh and purple arrows indicating torsional loading. The text 'MAIN MENU JOBS' is displayed in the center of the main window. The bottom status bar shows the command prompt with the following text: 'File is mentat, version unknown. Model file C:\Users\manto\Documents\201819_esami\PAOM\20190328_PAOM_lab\torsione_rev00.mud opened for reading. Ready Model C:\Users\manto\Documents\201819_esami\PAOM\20190328_PAOM_lab\torsione_rev00 opened. Command > *dynamic_model_off Command > |'.

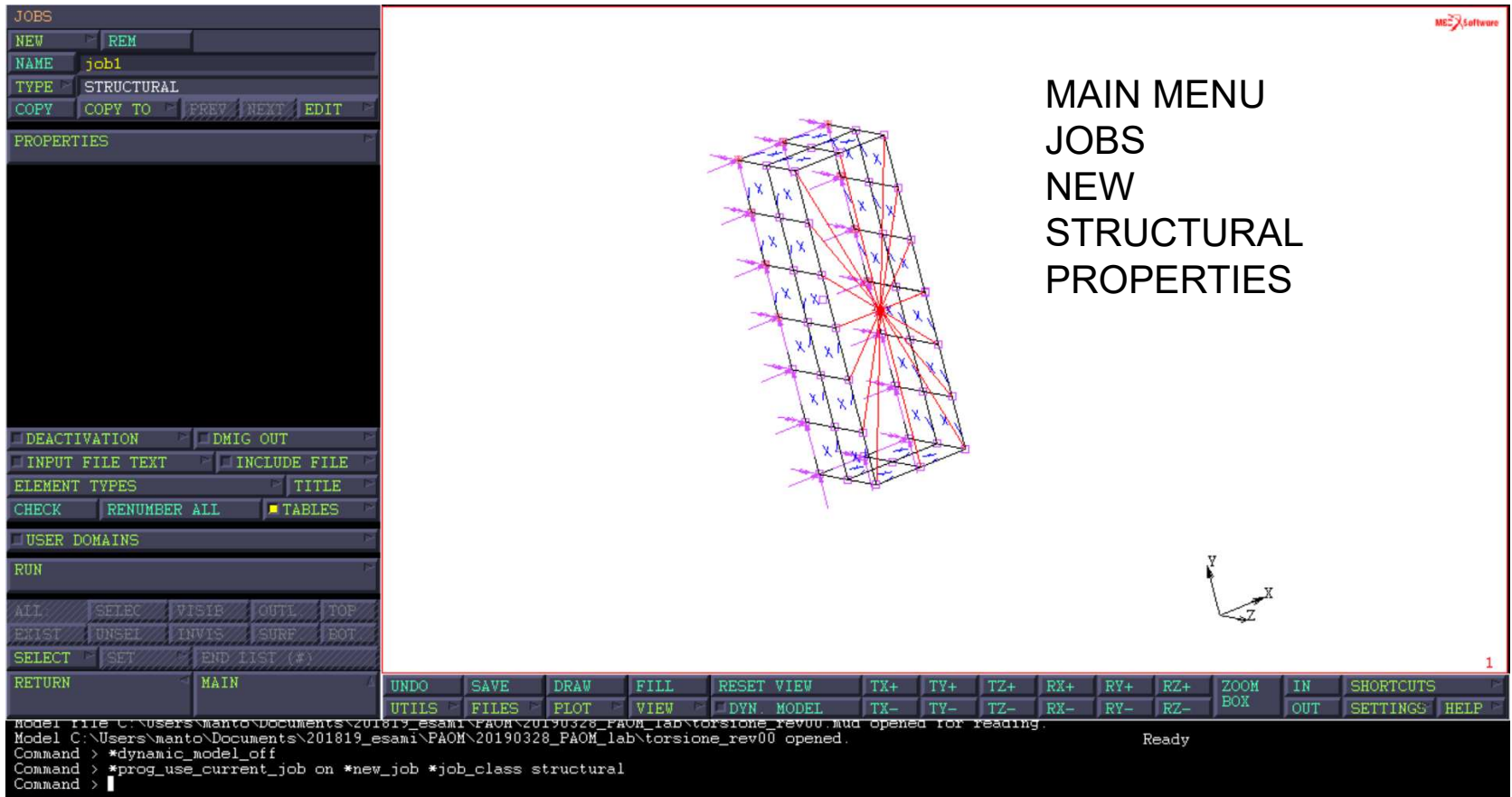
Thin-walled profile in torsion

JOBS

The screenshot displays the ANSYS software interface. On the left, the 'JOBS' menu is open, showing a hierarchical list of options. A red arrow points to the 'NEW' option, and another red arrow points to the 'STRUCTURAL' option. The main window shows a 3D model of a thin-walled profile under torsion, with a coordinate system (X, Y, Z) and a red arrow indicating the direction of the applied torque. The text 'MAIN MENU', 'JOBS', 'NEW', and 'STRUCTURAL' is overlaid on the right side of the main window. The bottom status bar shows 'Ready' and various utility buttons like 'UNDO', 'SAVE', 'DRAW', 'FILL', 'RESET VIEW', 'TX+', 'TY+', 'TZ+', 'RX+', 'RY+', 'RZ+', 'ZOOM BOX', 'IN', 'SHORTCUTS', 'OUT', 'SETTINGS', and 'HELP'.

Thin-walled profile in torsion

JOBS



The screenshot displays the software interface for setting up a torsion analysis on a thin-walled profile. On the left, the 'JOBS' panel is visible, with red arrows pointing to the 'NEW' button and the 'PROPERTIES' section. The main window shows a 3D model of a rectangular thin-walled profile with a mesh and applied loads. A coordinate system (X, Y, Z) is shown in the bottom right. The main menu is listed on the right side of the window.

MAIN MENU
JOBS
NEW
STRUCTURAL
PROPERTIES

Model File C:\Users\manto\Documents\201819_esami\PAOM\20190328_PAOM_lab\torsione_rev00.mud opened for Reading.
Model C:\Users\manto\Documents\201819_esami\PAOM\20190328_PAOM_lab\torsione_rev00 opened.
Command > *dynamic_model_off
Command > *prog_use_current_job on *new_job *job_class structural
Command > |

Thin-walled profile in torsion

JOBS

STRUCTURAL JOB PROPERTIES

LINEAR ELASTIC ANALYSIS

LOADCASES

INITIAL LOADS

INERTIA RELIEF

CONTACT CONTROL

MESH ADAPTIVITY

ACTIVE CRACKS

CRACK INITIATORS

DESIGN

CYCLIC SYMMETRY

GLOBAL-LOCAL

STEADY STATE ROLLING

MAP TEMPERATURE

MODEL SECTIONS

ANALYSIS OPTIONS

JOB RESULTS

JOB PARAMETERS

ANALYSIS DIMENSION

▼ 3-D

Command > *remove_current_job
Command > *prog_use_current_job on *new_job *job_class structural
Command > *remove_current_job
Command > *prog_use_current_job on *new_job *job_class structural

Thin-walled profile in torsion

JOBS

MAIN MENU
JOBS
NEW
STRUCTURAL
PROPERTIES
 INITIAL LOADS
 SKEW_SYMM_XY
 ROTATION_Z
OK

```
Command > *remove_current_job
Command > *prog_use_current_job on *new_job *job_class structural
Command > *remove_current_job
Command > *prog_use_current_job on *new_job *job_class structural
```

Thin-walled profile in torsion

JOBS

The screenshot shows the ANSYS Workbench Jobs dialog box. The 'JOBS' panel on the left shows 'job1' as the name and 'STRUCTURAL' as the type. The 'STRUCTURAL JOB PROPERTIES' panel is active, showing 'LINEAR ELASTIC ANALYSIS' selected. Under 'LOADCASES', there are 'SELECTED' and 'CLEAR' buttons. The 'AVAILABLE' section is empty. The 'INITIAL LOADS' option is checked and highlighted with a red box. Other options include 'DESIGN', 'CYCLIC SYMMETRY', 'GLOBAL-LOCAL', 'STEADY STATE ROLLING', 'MAP TEMPERATURE', and 'MODEL SECTIONS'. The 'ANALYSIS OPTIONS' section is expanded, showing 'JOB RESULTS', 'JOB PARAMETERS', and 'ANALYSIS DIMENSION' (set to '3-D'). A red arrow points to the 'JOB RESULTS' option. The 'COMMAND' window at the bottom shows the following commands:

```
Command > *remove_current_job
Command > *prog_use_current_job on *new_job *job_class structural
Command > *remove_current_job
Command > *prog_use_current_job on *new_job *job_class structural
Command >
```

The status bar at the bottom right shows 'Ready' and a 'HELP' button.

MAIN MENU
JOBS
NEW
STRUCTURAL
PROPERTIES
 INITIAL LOADS
JOB RESULTS

Thin-walled profile in torsion

JOBS

JOB RESULTS

POST FILE: BINARY
DEFAULT STYLE: INCREMENT FREQUENCY: 1
OUTPUT FILE: REBAR VERIFICATION
FLOWLINES: TRACKING
STATUS FILE: FORCE BALANCE
ADDITIONAL CONTACT FILES: I-DEAS, HYPERMESH, ADAMS

SELECTED ELEMENT QUANTITIES	LAYERS
Stress in Preferred Sys	OUT & MID
Equivalent Von Mises Stress	MAX & MIN
1st Element Orientation Vector	DEFAULT
2nd Element Orientation Vector	DEFAULT

AVAILABLE ELEMENT TENSORS

- Stress
- Stress in Preferred Sys
- Global Stress
- Cauchy Stress
- Cauchy Stress in Preferred Sys
- Global 2nd Piola-Kirchhoff Rebar Stress

AVAILABLE ELEMENT SCALARS

- Equivalent Von Mises Stress
- Mean Normal Stress
- Equivalent Cauchy Stress
- Total Strain Energy Density
- Equivalent Elastic Strain
- Elastic Strain Energy Density

ELEMENT RESULTS: ALL POINTS, CENTROID

SELECTED NODAL QUANTITIES: DEFAULT, CUSTOM

CONTACT GLUE FORCES: INCLUDE, EXCLUDE

ITERATIVE RESULTS: OFF

```
Enter job post element quantity : *add_post_var elem_orient_1
Enter add job post variable : *add_post_var elem_orient_2
Enter add job post variable :
```

JOB RESULTS

AVAILABLE ELEMENT TENSOR

- Stress in Preferred Sys (OUT & MID)

AVAILABLE ELEMENT SCALAR

- Equivalent Von Mises Stress (MAX & MIN)
- 1st Element Orientation Vector (DEFAULT)
- 2nd Element Orientation Vector (DEFAULT)

Thin-walled profile in torsion

JOBS

JOB RESULTS

...

SELECTED NODAL QUANTITIES

CUSTOM

AVAILABLE NODAL QUANTITIES

- Displacement
- Rotation
- Reaction Force
- Reaction Moment
- Tying Force
- Tying Moment

OK

JOB RESULTS

POST FILE BINARY OUTPUT FILE REBAR VERIFICATION

▼ DEFAULT STYLE INCREMENT FREQUENCY 1 FLOWLINES TRACKING

STATUS FILE FORCE BALANCE

SELECTED ELEMENT QUANTITIES LAYERS

<input checked="" type="checkbox"/> Stress in Preferred Sys	▼ OUT & MID	<input type="button" value="CLR"/>
<input checked="" type="checkbox"/> Equivalent Von Mises Stress	▼ MAX & MIN	<input type="button" value="CLR"/>
<input checked="" type="checkbox"/> 1st Element Orientation Vector	▼ DEFAULT	<input type="button" value="CLR"/>
<input checked="" type="checkbox"/> 2nd Element Orientation Vector	▼ DEFAULT	<input type="button" value="CLR"/>

ELEMENT RESULTS ALL POINTS CENTROID

SELECTED NODAL QUANTITIES DEFAULT CUSTOM

<input checked="" type="checkbox"/> Displacement	<input type="button" value="▲"/>	<input type="button" value="▼"/>
<input checked="" type="checkbox"/> Rotation	<input type="button" value="▲"/>	<input type="button" value="▼"/>
<input checked="" type="checkbox"/> Reaction Force	<input type="button" value="▲"/>	<input type="button" value="▼"/>
<input checked="" type="checkbox"/> Reaction Moment	<input type="button" value="▲"/>	<input type="button" value="▼"/>
<input checked="" type="checkbox"/> Tying Force	<input type="button" value="▲"/>	<input type="button" value="▼"/>

CONTACT GLUE FORCES INCLUDE EXCLUDE

ITERATIVE RESULTS ▼ OFF

AVAILABLE NODAL QUANTITIES

<input checked="" type="checkbox"/> Displacement	<input type="button" value="▲"/>	<input type="button" value="▼"/>
<input checked="" type="checkbox"/> Rotation	<input type="button" value="▲"/>	<input type="button" value="▼"/>
<input checked="" type="checkbox"/> Total Displacement	<input type="button" value="▲"/>	<input type="button" value="▼"/>
<input type="checkbox"/> External Force	<input type="button" value="▲"/>	<input type="button" value="▼"/>
<input type="checkbox"/> External Moment	<input type="button" value="▲"/>	<input type="button" value="▼"/>
<input checked="" type="checkbox"/> Reaction Force	<input type="button" value="▲"/>	<input type="button" value="▼"/>

Enter add job post nodal quantity : *add_post_nodal_quantity Ty_Force
Enter add job post nodal quantity : *add_post_nodal_quantity Ty_Moment

Thin-walled profile in torsion

JOBS

The screenshot shows the ANSYS Workbench Jobs dialog box. The 'JOBS' panel on the left shows a job named 'job1' of type 'STRUCTURAL'. The 'STRUCTURAL JOB PROPERTIES' panel is active, showing 'LINEAR ELASTIC ANALYSIS' selected under 'LOADCASES'. The 'ANALYSIS DIMENSION' is set to '3-D'. The 'OK' button is highlighted with a yellow box. A red arrow points to the 'ANALYSIS DIMENSION' dropdown menu, which is currently set to '3-D'. The 'COMMAND' window at the bottom shows the following commands:

```
Command > *remove_current_job  
Command > *prog_use_current_job on *new_job *job_class structural  
Command > *remove_current_job  
Command > *prog_use_current_job on *new_job *job_class structural  
Command > |
```

MAIN MENU

JOBS

NEW

STRUCTURAL

PROPERTIES

INITIAL LOADS


ANALYSIS DIMENSION

3-D

OK

Thin-walled profile in torsion

JOBS



The screenshot displays the software interface for a finite element analysis. On the left, a panel titled "JOBS" contains fields for "NAME" (job1), "TYPE" (STRUCTURAL), and "COPY" options. Below this is a "PROPERTIES" section and a "RUN" section with various options like "ALL", "SELEC", "VISIB", "OUTL", "TOP", "EXIST", "UNSEL", "INVIS", "SURF", "BOT", "SELECT", "SET", "END LIST (*)", and "RETURN". A red arrow points to the "ELEMENT TYPES" field. The main window shows a 3D model of a thin-walled profile under torsion, with a coordinate system (X, Y, Z) at the bottom right. The "MAIN MENU" is visible on the right side of the main window, listing "JOBS", "NEW", "STRUCTURAL", and "ELEMENT TYPES". The command line at the bottom shows the following commands:

```
Model File C:\Users\manto\Documents\201819_esami\PAOM\20190328_PAOM_lab\torsione_rev00.mud opened for Reading.  
Model C:\Users\manto\Documents\201819_esami\PAOM\20190328_PAOM_lab\torsione_rev00 opened.  
Command > *dynamic_model_off  
Command > *prog_use_current_job on *new_job *job_class structural  
Command > |
```

Thin-walled profile in torsion

JOBS

MAIN MENU
JOBS
NEW
STRUCTURAL
ELEMENT TYPES
SHELL/MEMBRANE
THICK SHELL
75
OK

	TRIA		QUAD	
	3	6	4	8
THICK SHELL			75	22
THICK SHELL - REDUCED INTEGRATION			140	
THIN SHELL	138	49	139	72
MEMBRANE	158	200	18	30
MEMBRANE REBAR			147	148
SHEAR PANEL			68	

Thin-walled profile in torsion

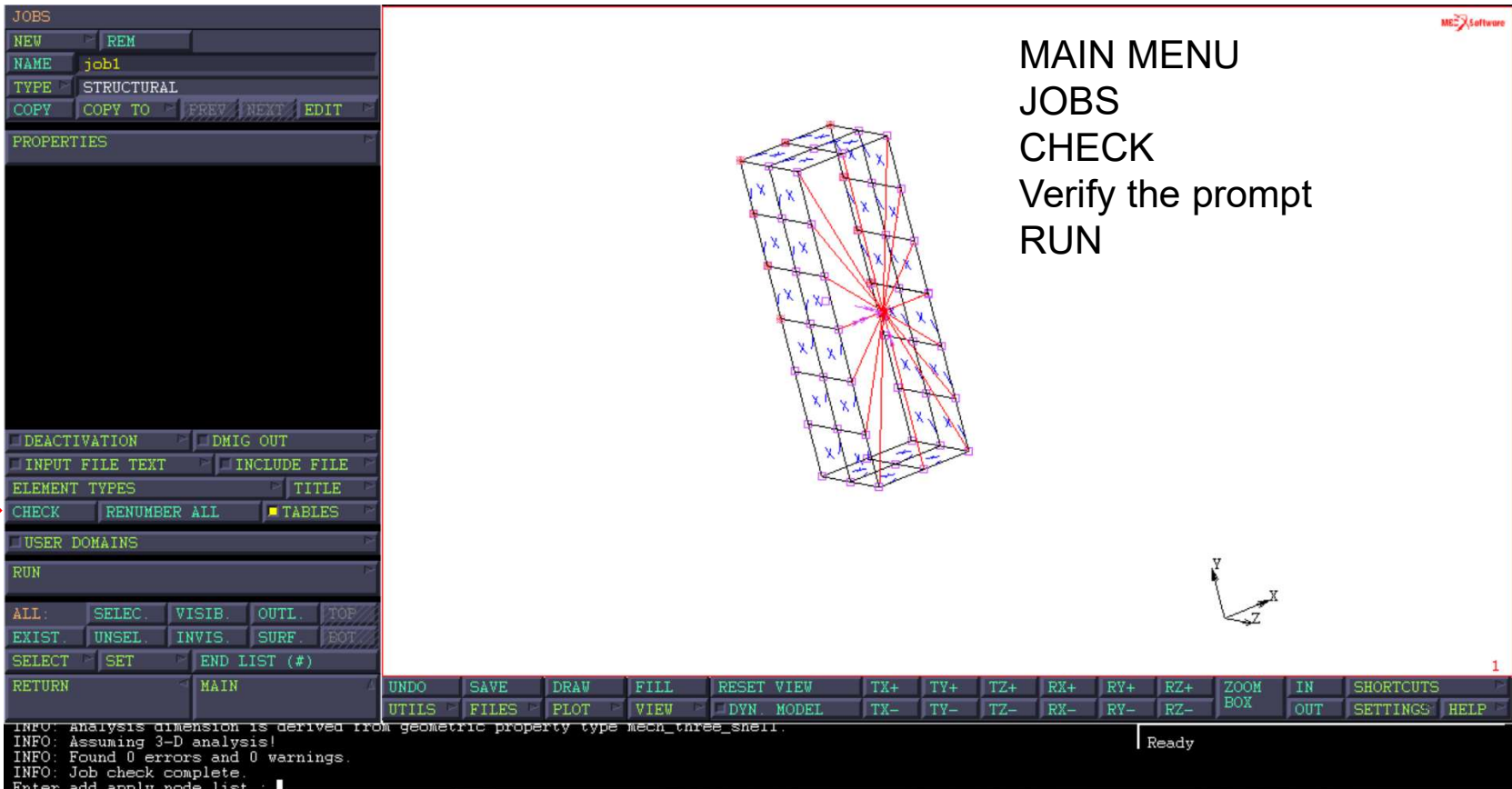
JOBS



The screenshot displays the ANSYS software interface. On the left, the 'ELEMENT TYPES' menu is open, showing the following options: ANALYSIS CLASS (STRUCTURAL), ANALYSIS DIMENSION (3-D), SOLID, SOLID COMPOSITE/GASKET, SOLID SHELL, INTERFACE, SHELL/MEMBRANE, TRUSS/BEAM, and MISCELLANEOUS. A red arrow points to the 'SHELL/MEMBRANE' option. The main window shows a 3D model of a thin-walled profile in torsion, with a coordinate system (X, Y, Z) visible. The model is a rectangular prism with a grid of nodes and elements, and a red arrow indicating the direction of torsion. The 'MAIN MENU' is visible on the right side of the window, listing: MAIN MENU, JOBS, NEW, STRUCTURAL, ELEMENT TYPES, SHELL/MEMBRANE, THICK SHELL, 75, OK, and ALL:EXIST. The bottom status bar shows the command prompt with the following text: 'Enter add apply mode list : *element_type 75', 'Enter element list : all_existing', 'Number of unchanged elements: 0', 'Command > *element_type 75', and 'Enter element list :'. The status bar also shows 'Ready' and a small '1' in the bottom right corner.

Thin-walled profile in torsion

JOBS



JOBS

NEW REM

NAME job1

TYPE STRUCTURAL

COPY COPY TO PREV NEXT EDIT

PROPERTIES

DEACTIVATION DMIG OUT

INPUT FILE TEXT INCLUDE FILE

ELEMENT TYPES TITLE

CHECK RENUMBER ALL TABLES

USER DOMAINS

RUN

ALL: SELEC VISIB OUTL TOP

EXIST UNSEL INVIS SURF BOT

SELECT SET END LIST (#)

RETURN MAIN

UNDO SAVE DRAW FILL RESET VIEW TX+ TY+ TZ+ RX+ RV+ RZ+ ZOOM IN SHORTCUTS

UTILS FILES PLOT VIEW DYN. MODEL TX- TY- TZ- RX- RV- RZ- ZOOM BOX OUT SETTINGS HELP

INFO: analysis dimension is derived from geometric property type mech_three_shell.
INFO: Assuming 3-D analysis!
INFO: Found 0 errors and 0 warnings.
INFO: Job check complete.
Enter add apply node list :

Ready

MAIN MENU
JOBS
CHECK
Verify the prompt
RUN

Thin-walled profile in torsion

JOBS

MAIN MENU
JOBS
CHECK
Verify the prompt
RUN
SUBMIT
EXIT NUMBER 3004

INFO: Assuming 3-D analysis!
INFO: Found 0 errors and 0 warnings.
C:\MSC\1_SOF\MARC_C\1\20131\1.0\MENTAT\1.1\bin\submit1.bat marc default 1 1 - - torsione_rev00_job1 - - - - 0 n n - -
Job job1 submitted!
Command > |

Thin-walled profile in torsion

JOBS

RUN JOB

USER SUBROUTINE FILE

PARALLELIZATION/GPU: No DDM
1 Assembly/Recovery Thread
1 Solver Thread
No GPU(s)

TITLE: STYLE: TABLE-DRIVEN SAVE MODEL

SUBMIT (1) ADVANCED JOB SUBMISSION

UPDATE MONITOR KILL

STATUS: Complete

CURRENT INCREMENT (CYCLE): 0 (1)

SINGULARITY RATIO: 7.7981e-017

CONVERGENCE RATIO: 0

ANALYSIS TIME: 0

WALL TIME: 11

TOTAL	CYCLES	1	CUT BACKS	0
	SEPARATIONS	0	REMESHES	0

EXIT NUMBER: 3004 EXIT MESSAGE

EDIT: OUTPUT FILE LOG FILE STATUS FILE ANY FILE

OPEN POST FILE (RESULTS MENU)

RESET OK

This is a successful completion to a Marc simulation, indicating that no additional incremental data was found and that the analysis is complete.

Marc 2013.1.0

Exit number 3004

Agenda

Finite Element Modelling

MSC Marc Mentat

Thin-walled profile in torsion

- The problem
- Mesh generation
- Geometric properties
- Material properties
- Links: RBE2
- Boundary conditions
- Jobs
- Results

Thin-walled profile in torsion UPDATE

Mesh convergence

References

Thin-walled profile in torsion

RESULTS

OPEN POST FILE (RESULTS MENU)
DEFORMED SHAPE
STYLE: DEFORMED AND ORIGINAL
CLOSE

Inc: 0
Time: 0.000e+000

POSTPROCESSING RESULTS

POST FILE

C:\Users\manto\Documents\201819_esam:

OPEN DEFAULT OPEN CLOSE

MONITOR SCAN

REWIND PREV NEXT LAST

SKIP TO INC SKIP INCS

DEFORMED SHAPE SETTINGS

STYLE DEFORMED & ORIGINAL

SCALAR PLOT SETTINGS

OFF CONTOUR BANDS

CONTOUR CENT CONTOUR LINES

SYMBOLS NUMERICS

ISO-SURFACES CUTTING PLANES

BEAM CONTOURS BEAM VALUES

SCALAR Displacement X

SAMPLE POINTS PATH PLOT

HISTORY PLOT TRACK PLOT

DESIGN PLOT GLOBAL VARIABLES

TOOLS

MORE

ALL SELEC VISIB OUTL TOP

EXIST UNSEI INVIS SURF BOT

SELECT SET END LIST (#)

RETURN MAIN

Command > *set deformed both
Command > *dynamic_model_on
Command > *reset_view
Command > *fill_view
Command >

job1

Ready

RUN JOB

USER SUBROUTINE FILE

PARALLELIZATION/GPU No DDH
1 Assembly/Recovery Thread
1 Solver Thread
No GPU(s)

TITLE STYLE TABLE-DRIVEN SAVE MODEL

SUBMIT (1) ADVANCED JOB SUBMISSION

UPDATE MONITOR KILL

STATUS Complete

CURRENT INCREMENT (CYCLE) 0 (1)

SINGULARITY RATIO 7.7981e-017

CONVERGENCE RATIO 0

ANALYSIS TIME 0

WALL TIME 11

TOTAL CYCLES 1 CUT BACKS 0

SEPARATIONS 0 REMESHES 0

EXIT NUMBER 3004 EXIT MESSAGE

EDIT OUTPUT FILE LOG FILE STATUS FILE ANY FILE

OPEN POST FILE (RESULTS MENU)

RESET OK

Agenda

Finite Element Modelling

MSC Marc Mentat

Thin-walled profile in torsion

Thin-walled profile in torsion UPDATE

- Boundary conditions
- Jobs

Mesh convergence

References

Thin-walled profile in torsion

BOUNDARY CONDITIONS

BOUNDARY CONDITIONS

ANALYSIS CLASS
STRUCTURAL

NEW REM
NAME positioning_z
TYPE fixed_displacement
COPY PREV NEXT EDIT

PROPERTIES

PLOT SETTINGS
 DRAW BOUNDARY CONDS ON MESH
 ID BOUNDARY CONDS
ARROW PLOT SETTINGS

MERGE DUPLICATE BOUND CONDS
REMOVE ALL BOUND CONDS

TABLES TRANSFORMATIONS

NODES ADD REM 0
POINTS ADD REM 0
CURVES ADD REM 0
SURFACES ADD REM 0

ALL SELEC VISIB OUTI TOP
EXIST UNSEL INVIS SURF BOT
SELECT SET END LIST (#)
RETURN MAIN

```

Model saved to C:\Users\manto\documents\201
Command > *new_apply *apply_type fixed_disp
Command > *apply_name positioning_z
Command > *apply_dof z *apply_dof_value z
Enter value for 'z' : 1
    
```

FIXED DISPLACEMENT

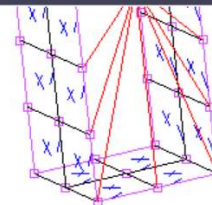
METHOD ENTERED VALUES

REFERENCE POSITION POSITION AT ACTIVATION OF BC

TIME DEPENDENCE TABLES

DISPLACEMENT X
 DISPLACEMENT Y
 DISPLACEMENT Z 0 TABLE
 ROTATION X
 ROTATION Y
 ROTATION Z

CLEAR OK



BOUNDARY CONDITIONS
NEW

STRUCTURAL
FIXED DISPLACEMENT
 DISPLACEMENT Z

NODES: ADD

Select only one node of the model,
e.g. at the skew symm-plane (5)

1

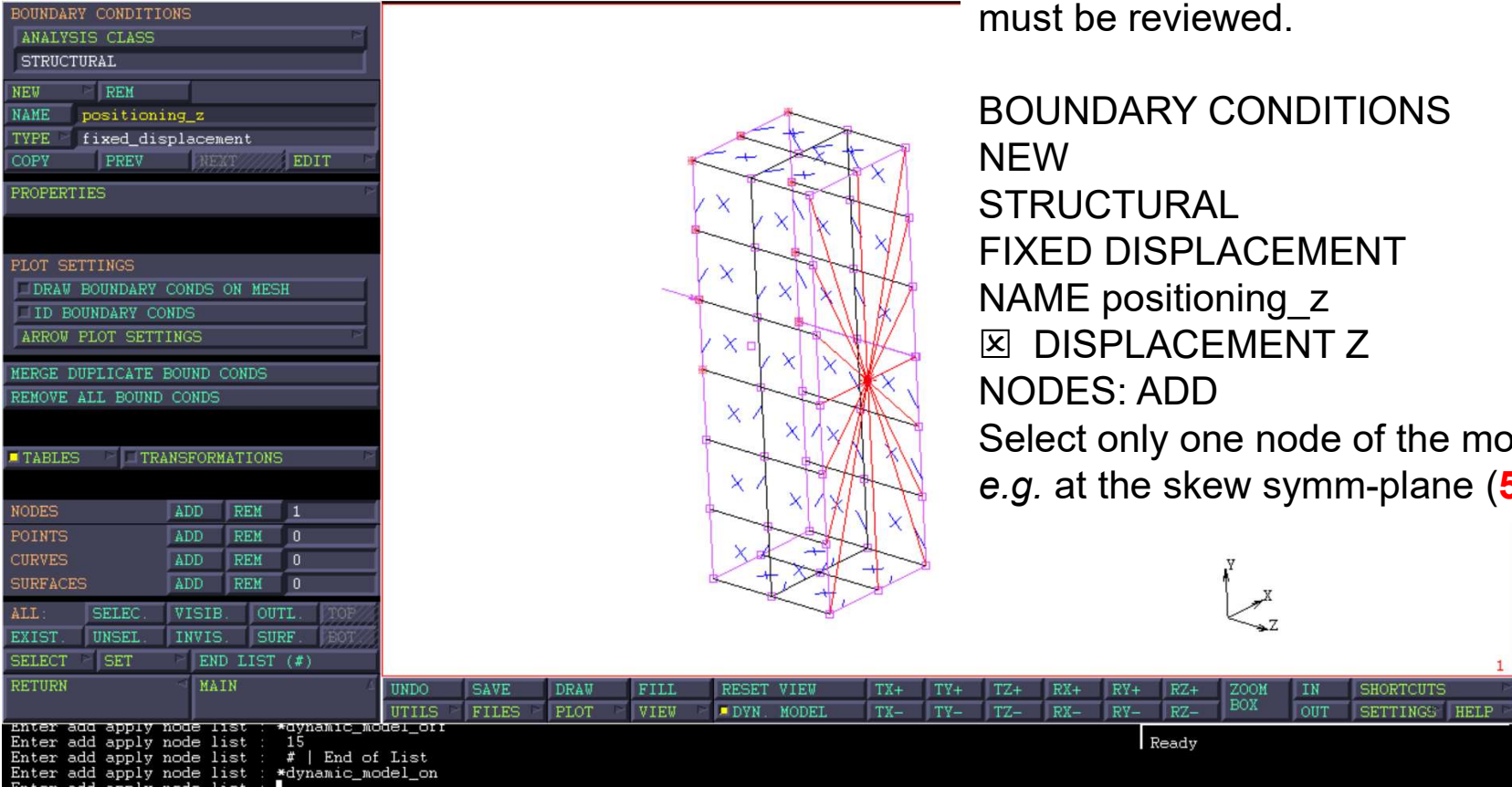
RZ+ ZOOM IN SHORTCUTS
RZ- BOX OUT SETTINGS HELP

ready

Thin-walled profile in torsion UPDATE

BOUNDARY CONDITIONS

The motion of the rigid body must be removed, therefore the BCs must be reviewed.



BOUNDARY CONDITIONS

ANALYSIS CLASS
STRUCTURAL

NEW REM

NAME positioning_z

TYPE fixed_displacement

COPY PREV NEXT EDIT

PROPERTIES

PLOT SETTINGS

DRAW BOUNDARY CONDS ON MESH

ID BOUNDARY CONDS

ARROW PLOT SETTINGS

MERGE DUPLICATE BOUND CONDS

REMOVE ALL BOUND CONDS

TABLES TRANSFORMATIONS

NODES	ADD	REM	1
POINTS	ADD	REM	0
CURVES	ADD	REM	0
SURFACES	ADD	REM	0

ALL: SELEC. VISIB. OUTL. TOP

EXIST. UNSEL. INVIS. SURF. BOT

SELECT SET END LIST (#)

RETURN MAIN

UNDO SAVE DRAW FILL RESET VIEW TX+ TY+ TZ+ RX+ RY+ RZ+ ZOOM IN SHORTCUTS

UTILS FILES PLOT VIEW DYN. MODEL TX- TY- TZ- RX- RY- RZ- ZOOM BOX OUT SETTINGS HELP

Ready

Enter add apply node list : *dynamic_model_on
Enter add apply node list : 15
Enter add apply node list : # | End of List
Enter add apply node list : *dynamic_model_on
Enter add apply node list :

BOUNDARY CONDITIONS

NEW

STRUCTURAL

FIXED DISPLACEMENT

NAME positioning_z

DISPLACEMENT Z

NODES: ADD

Select only one node of the model,
e.g. at the skew symm-plane (5)

Thin-walled profile in torsion UPDATE

JOBS

The jobs must be uploaded including the new BCs named *positioning_z*.

STRUCTURAL JOB PROPERTIES

SELECT INITIAL LOADS

BOUNDARY CONDITIONS CLEAR

<input checked="" type="checkbox"/> skew_symm_xy	fixed_displacement
<input checked="" type="checkbox"/> rotation_z	fixed_displacement
<input checked="" type="checkbox"/> positioning_z	fixed_displacement

INITIAL CONDITIONS CLEAR

OK

CHECK
RUN
SUBMIT

Ready

SELECTED INITIAL JOB

- skew_symm_xy
- rotation_z
- positioning_z

OK

All the BCs must be switched on. Then the model is ready to be calculated once again starting from the JOB menu:

CHECK
RUN
SUBMIT

Thin-walled profile in torsion UPDATE

JOBS

Now, we evaluate the model results, as seen before, from the menu:

OPEN POST FILE (RESULTS MENU)
DEFORMED SHAPE
STYLE: DEFORMED AND ORIGINAL

The screenshot displays the ANSYS Workbench Jobs panel. The 'RUN JOB' dialog box is open, showing the 'UPDATE' button highlighted with a red arrow. The 'OPEN POST FILE (RESULTS MENU)' option is highlighted with a yellow box. A red arrow points to the 'STATUS' section of the dialog, which shows 'Complete' and 'WALL TIME: 10'. The 'EXIT MESSAGE' field contains the text: 'ERROR: Job already running!' and 'ERROR: Job already running!'.

STATUS				
STATUS	Complete			
CURRENT INCREMENT (CYCLE)	0 (1)			
SINGULARITY RATIO	0.013275			
CONVERGENCE RATIO	0			
ANALYSIS TIME	0			
WALL TIME	10			
TOTAL	CYCLES	1	CUT BACKS	0
	SEPARATIONS	0	REMESHES	0
EXIT NUMBER	3004			
EXIT MESSAGE	ERROR: Job already running! ERROR: Job already running!			

Agenda

Finite Element Modelling

MSC Marc Mentat

Thin-walled profile in torsion

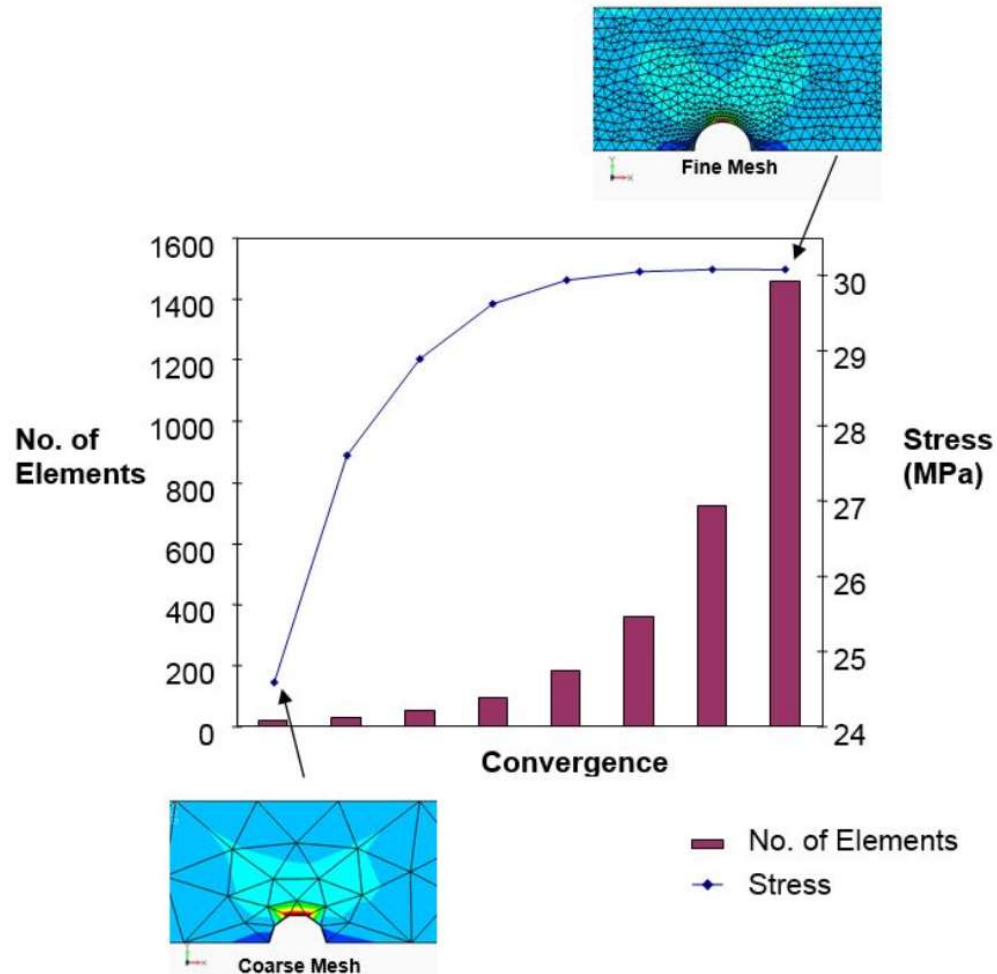
Thin-walled profile in torsion UPDATE

Mesh convergence

References

Thin-walled profile in torsion

MESH SIZE: CONVERGENCE



Agenda

Finite Element Modelling

MSC Marc Mentat

Thin-walled profile in torsion

Thin-walled profile in torsion UPDATE

Mesh convergence

References

References

LAB Marc Mentat files saved as:

torsione_rev01_labile.mud

torsione_rev01_labile.t16

torsione_rev01_nolabile.mud

torsione_rev01_nolabile.t16

thin_walled_profile_in_torsion.proc

«It is much easier to make measurements than to know exactly what you are measuring».

J. W. N. Sullivan



Sara Mantovani
Via Vivarelli, 10
41125, Modena, Italy
Mail: sara.mantovani@unimore.it
Mail: millechili@unimore.it
Phone: +39 059 2056280