UNIMORE
UNIVERSITÀ DEGLI STUDI DI MODENA E REGGIO EMILIA

Dipartimento di Ingegneria "Enzo Ferrari"

MILLE CHILI

Progettazione Assistita di Organi di Macchine

Sara Mantovani
sara.mantovani@unimore.it

Agenda

Goal

The geometry import

Mesh generation

Contact

References

Agenda

Goal

The geometry import

Mesh generation

Contact

References

GOAL

Connecting rod and bush: interference loadcase

The present model evaluates the contact pressure occuring in a connecting rod-bush assembly at the bush mounting stage for a radial interference fit equal to 0.03 mm.

This contact is a stationary contact. (increasing the load, the contact area remains constant).

The con-rod shank may be either I-shaped (the shank pockets are frontal) or H-shaped (the pockets are lateral). In the present model, the first one is considered.

Another difficult problem is the selection of the correct interference between eye and bush, that prevents an undesired bush loosening and the microslip between the bush and the small end that may produce fretting fatigue cracks.

I-shaped

H-shaped

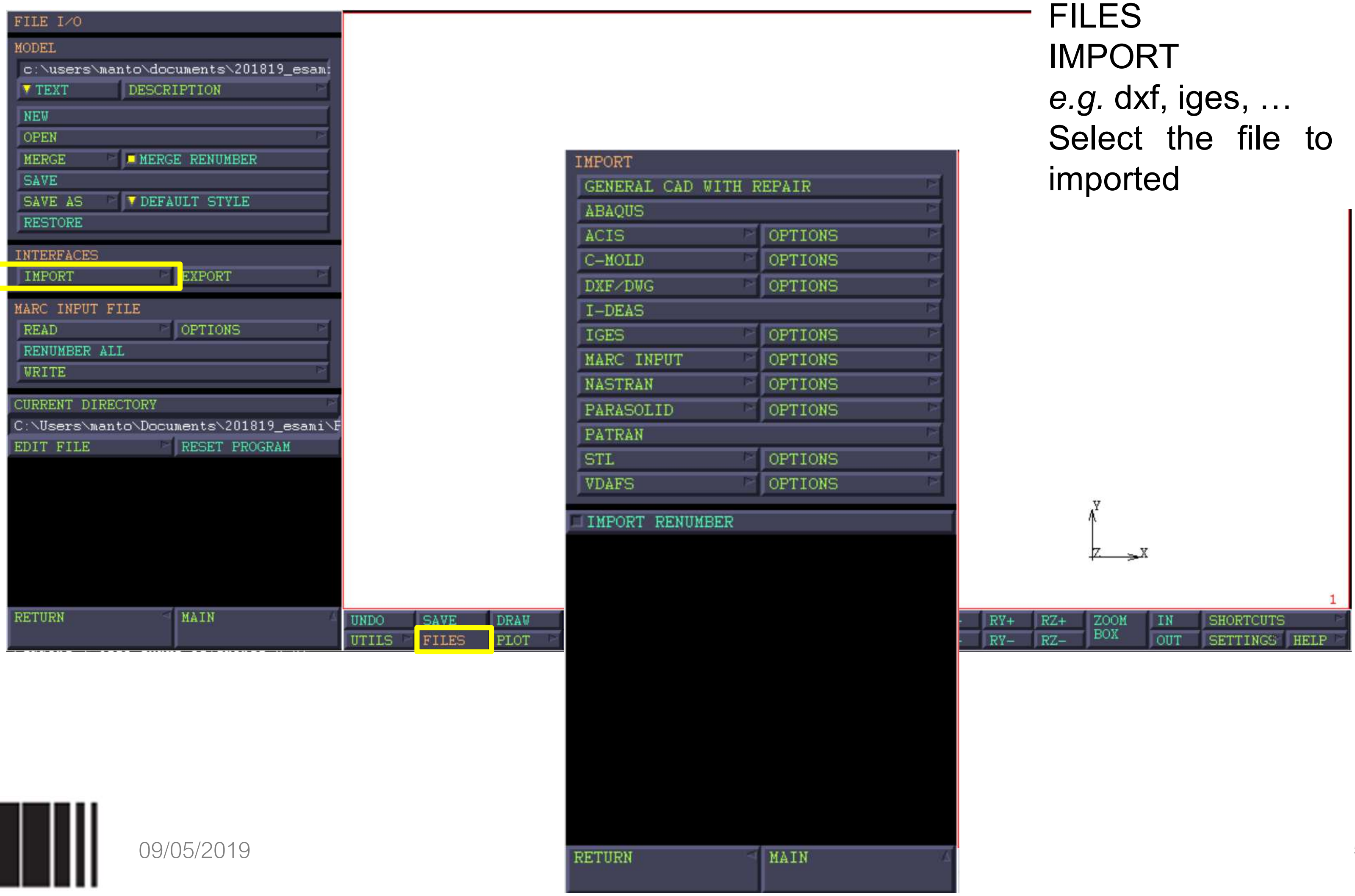
Bush

Bush loosening

The geometry import

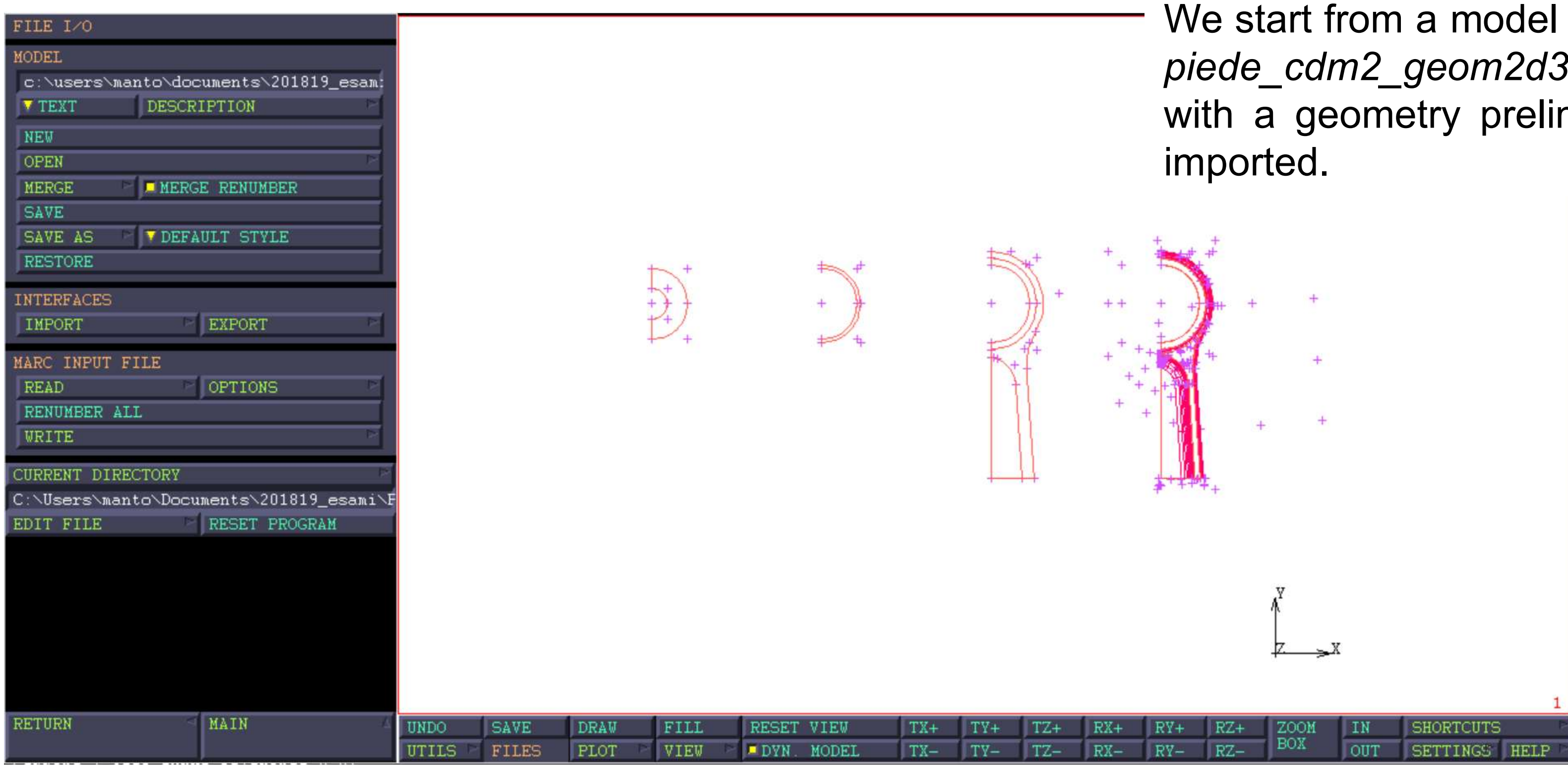
Connecting rod and bush: interference loadcase

MAIN MENU
FILES
IMPORT
e.g. dxf, iges, …
Select the file to be imported



The geometry import

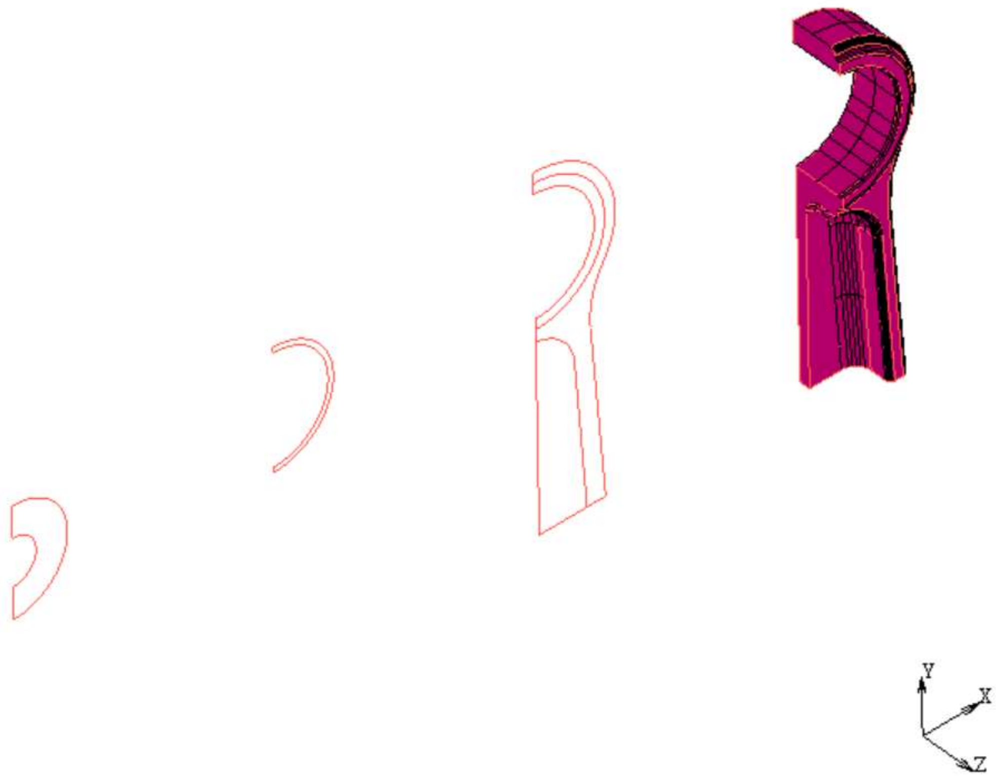
Connecting rod and bush: interference loadcase



We start from a model called *piede_cdm2_geom2d3d.mfd* with a geometry preliminary imported.

The geometry import

Connecting rod and bush: interference loadcase



We start from a model called *piede_cdm2_geom2d3d.mfd* with a geometry preliminary imported.

The model consists of:

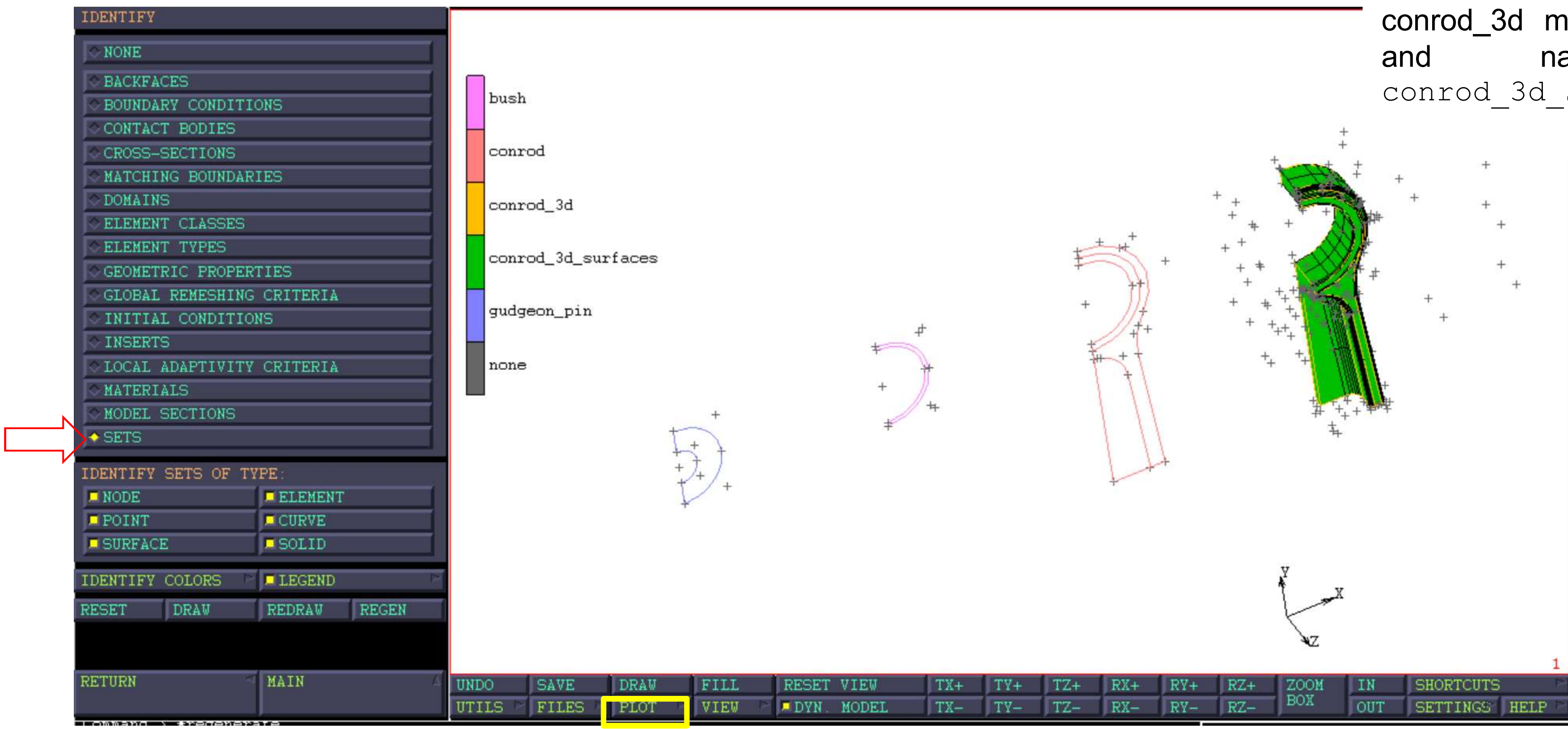
- Connecting rod 3D;
- Connecting rod 2D;
- Bush;
- Gudgeon pin.

The geometry import

Identify set of curves

PLOT
IDENTIFY
☒ SETS

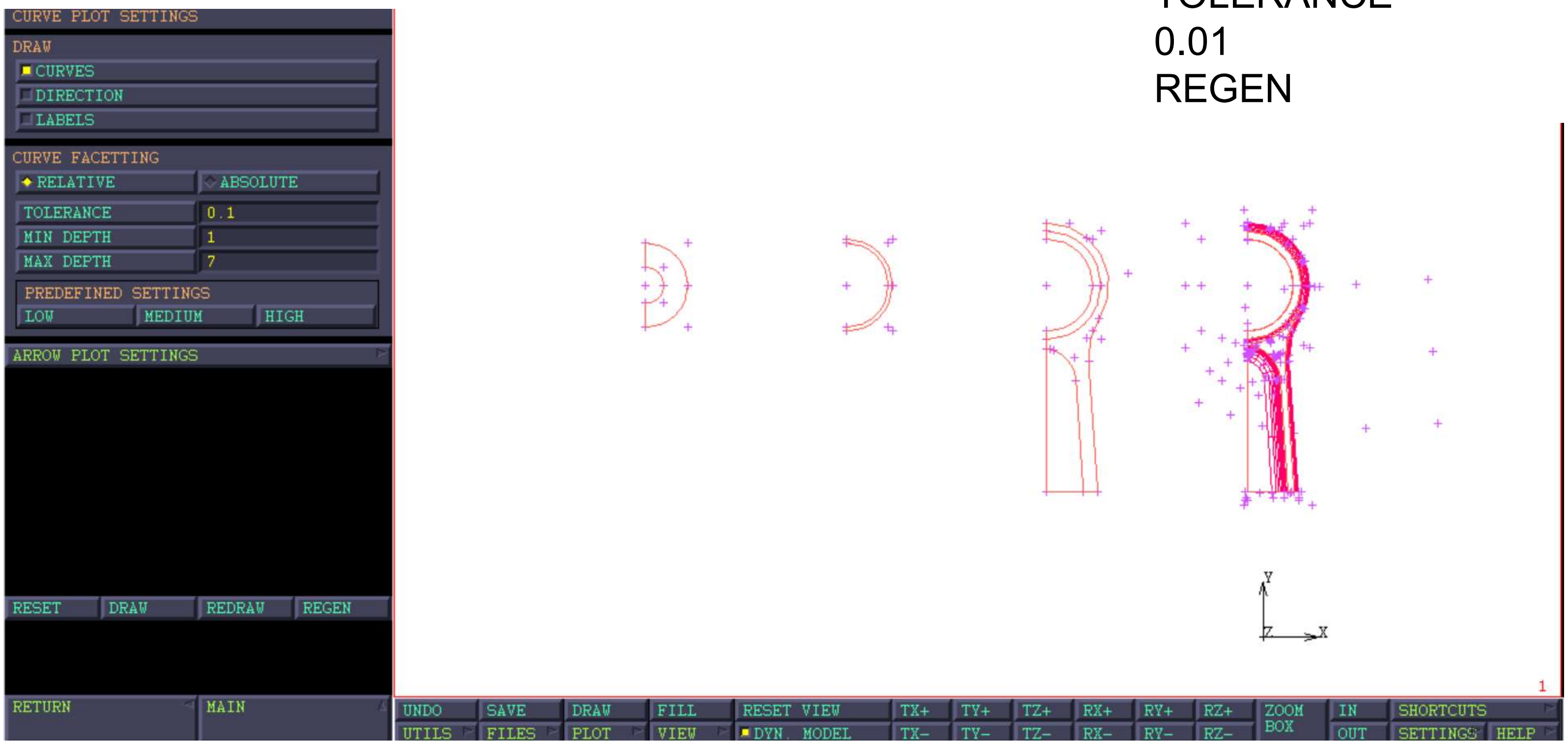
There are four sets of curves that store the curves referring to each component. *e.g.* bush, conrod, gudgeon_pin
The surface of the conrod_3d model is stored and named as `conrod_3d_surfaces`.



The geometry import

The curve plot

PLOT
CURVES
TOLERANCE
0.01
REGEN



The geometry import
The curve plot

◆ DIVISIONS

Menu: DIVISIONS

Description: Curve and Surface Plotting Controls

Curves and surfaces are represented graphically by linear approximations, referred to as facets.

The breakup is performed by recursively subdividing the curve or surface until a specified deviation tolerance distance is satisfied. This tolerance may be specified in absolute or relative terms.

The breakup of curves and surface into facets is controlled by the following settings:

Relative/Absolute:
Specifies whether the tolerance is in terms of an absolute distance, or is in terms of the distance relative to the length of the curve or area of the surface.

Tolerance:
The maximum allowed deviation in absolute or relative distance of the facet to the curve or surface.

Max Depth:
The maximum depth of recursion allowed when subdividing the facets. Subdivision will continue until the tolerance is satisfied, or the maximum recursion depth is reached.

Min Depth:
This specifies the minimum depth of recursion, and when set to nonzero values forces the curve or surface to be represented by a minimum number of facets, even though the tolerance may have been satisfied.

It is recommended that relative tolerances be used for the general case. Absolute tolerance may be used to advantage to minimize the drawing of small details in a model whose dimension is known.

For convenience, three predefined settings: *low*, *medium*, and *high* are provided. The low setting is designed to minimize drawing time, while the high setting is designed to provide an extremely accurate representation of the geometry. The default setting is medium.

PLOT
CURVES
TOLERANCE
0.01
REGEN

Agenda

Goal

The geometry import

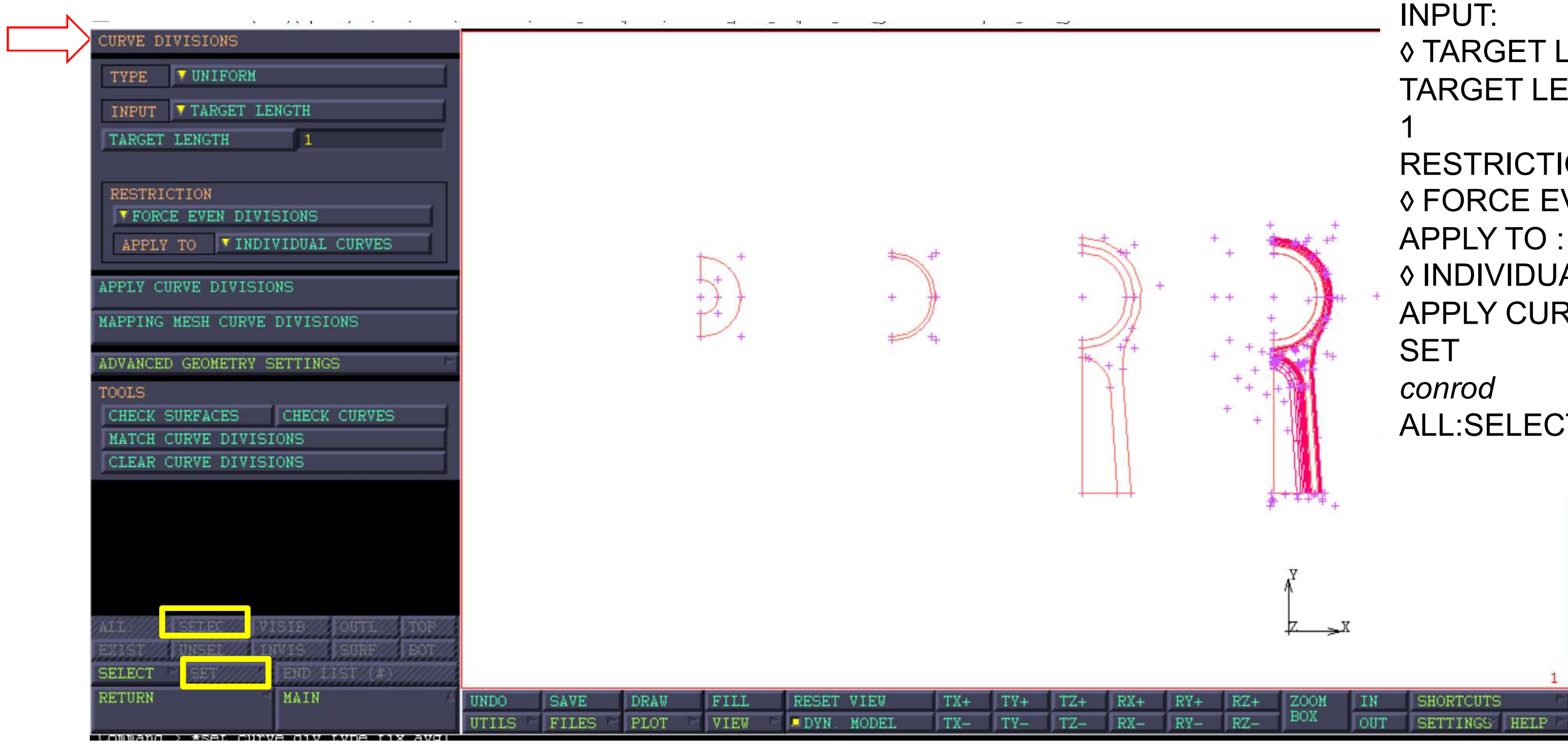
Mesh generation

Contact

References

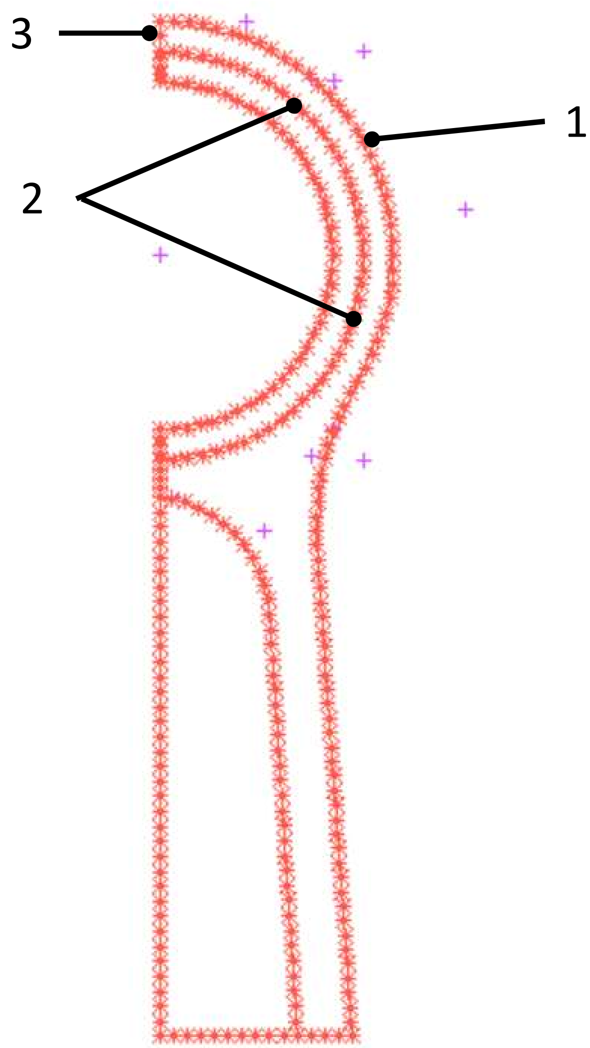
Mesh generation

Connecting rod 2D model



MAIN
MESH GENERATION
AUTOMESH
CURVE DIVISIONS
TYPE:
◊ UNIFORM
INPUT:
◊ TARGET LENGTH
TARGET LENGTH
1
RESTRICTION:
◊ FORCE EVEN DIVISION
APPLY TO :
◊ INDIVIDUAL CURVES
APPLY CURVE DIVISIONS
SET
conrod
ALL:SELECT

Mesh generation
Connecting rod 2D model



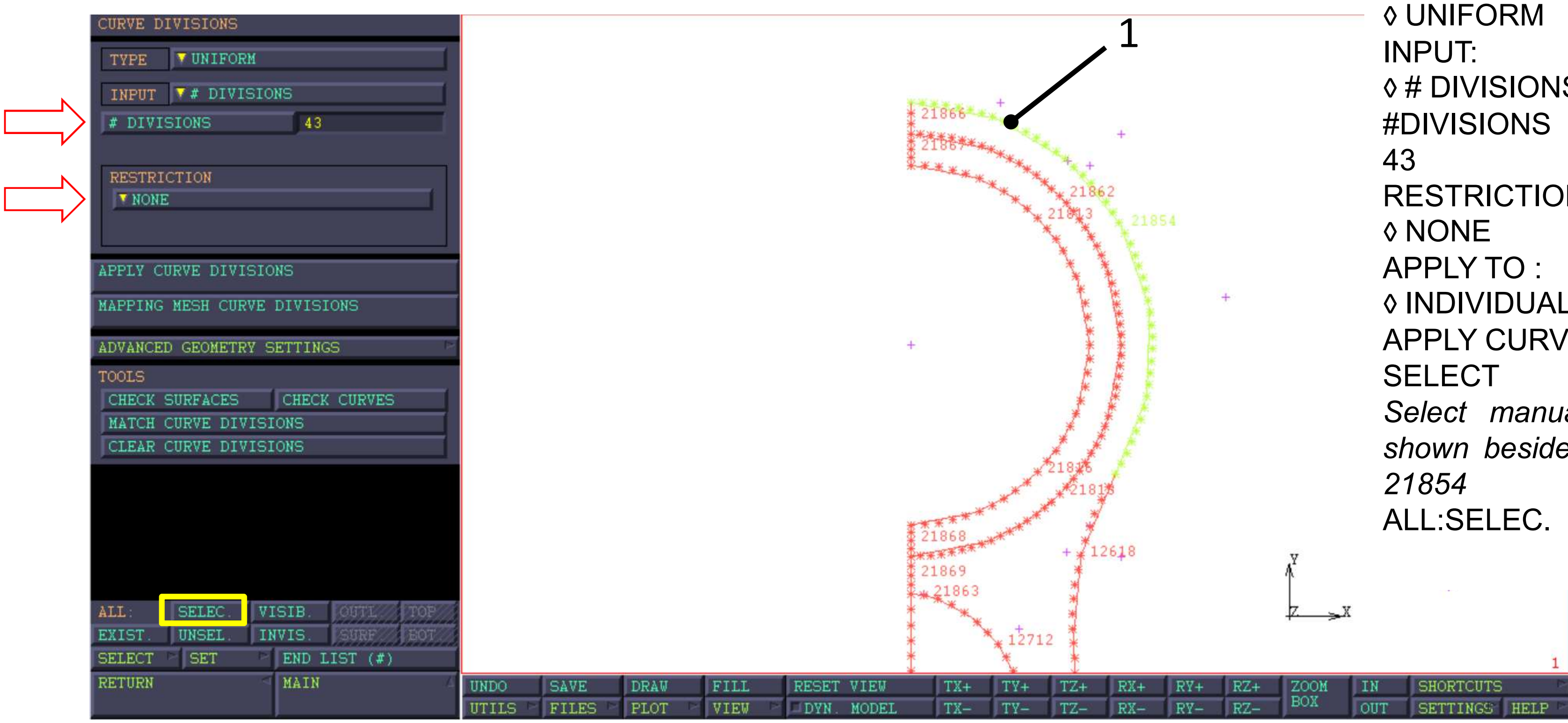
A detailed view of the connecting rod curve division.

The con-rod geometry consists of:
- the small end
- the frontal pocket at the shank

The mesh is too coarse to evaluate with a good accuracy the contact pressure between the components in contact.
Therefore, local adjustments of the mesh are performed at zone named:
1. Outer circumferencial edge at small end;
2. Middle circumferencial edges at small end;
3. Vertical edge at small end.

Mesh generation

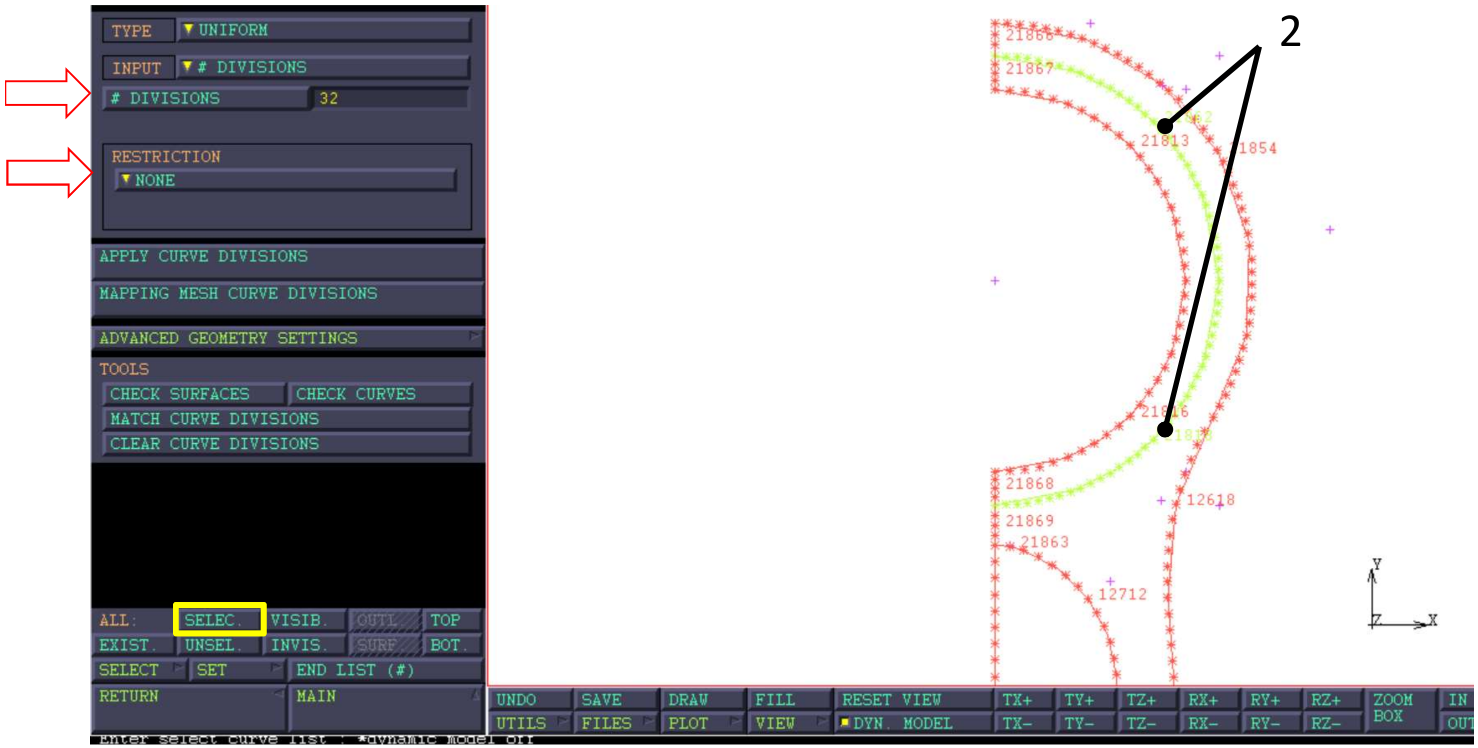
Connecting rod 2D model



MAIN
MESH GENERATION
AUTOMESH
CURVE DIVISIONS
TYPE:
◊ UNIFORM
INPUT:
◊ # DIVISIONS
#DIVISIONS
43
RESTRICTION:
◊ NONE
APPLY TO :
◊ INDIVIDUAL CURVES
APPLY CURVE DIVISIONS
SELECT
Select manually the curve shown beside with ID label: 21854
ALL:SELEC.

Mesh generation

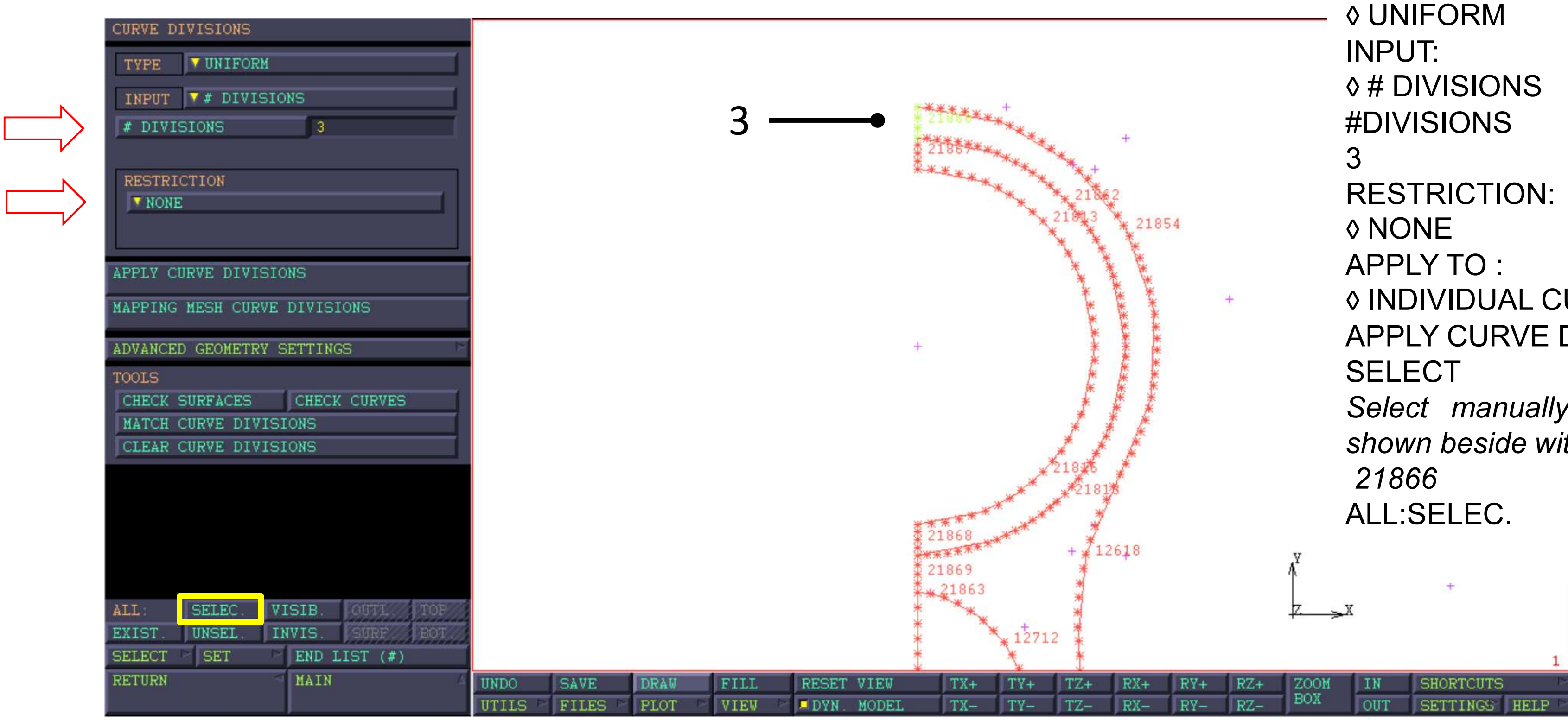
Connecting rod 2D model



MAIN
MESH GENERATION
AUTOMESH
CURVE DIVISIONS
TYPE:
◊ UNIFORM
INPUT:
◊ # DIVISIONS
#DIVISIONS
32
RESTRICTION:
◊ NONE
APPLY TO :
◊ INDIVIDUAL CURVES
APPLY CURVE DIVISIONS
SELECT
Select manually the curves shown beside with ID label:
21862
21818
ALL:SELEC.

The smalla end at the miiddle curves will be discretized by 64 circumferential divisions.

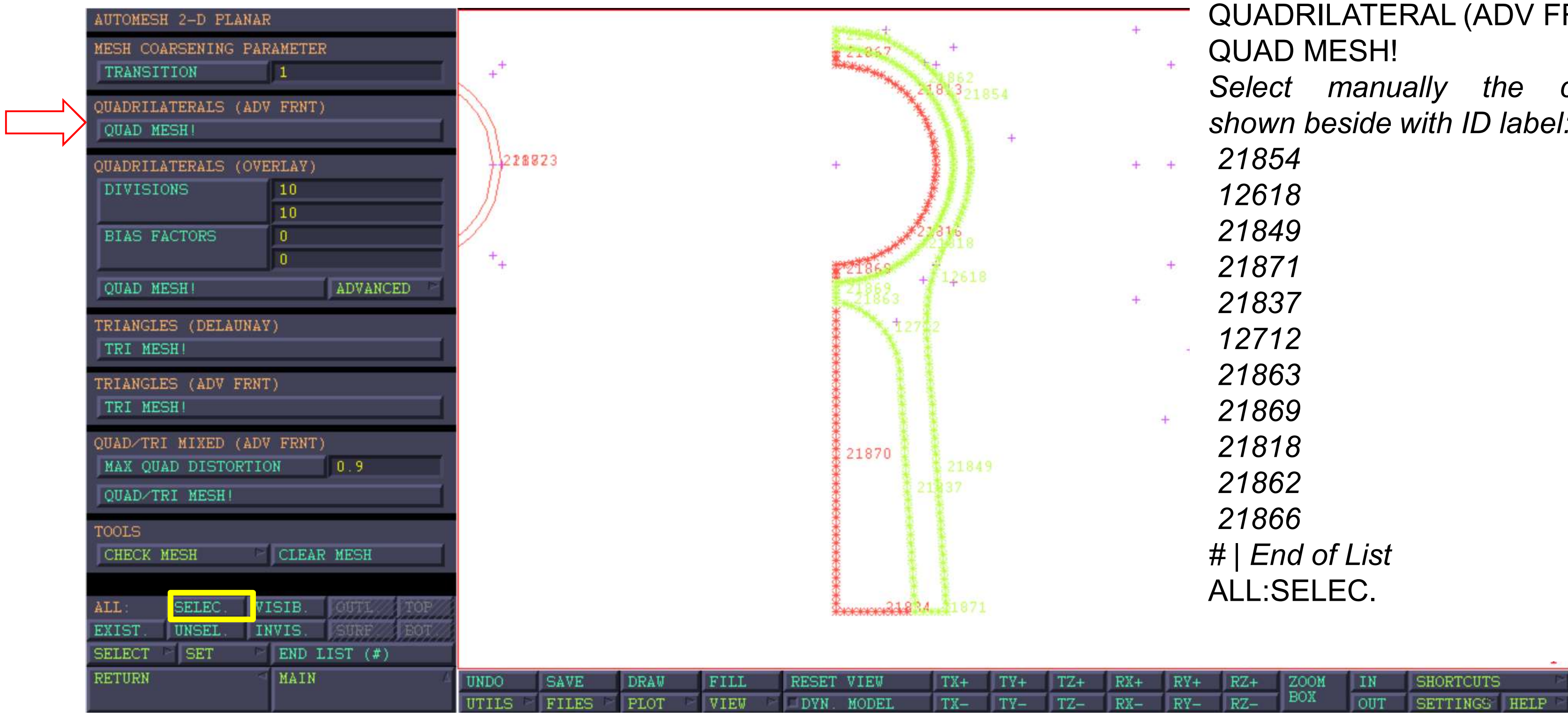
Mesh generation
Connecting rod 2D model



MAIN
MESH GENERATION
AUTOMESH
CURVE DIVISIONS
TYPE:
◊ UNIFORM
INPUT:
◊ # DIVISIONS
#DIVISIONS
3
RESTRICTION:
◊ NONE
APPLY TO :
◊ INDIVIDUAL CURVES
APPLY CURVE DIVISIONS
SELECT
Select manually the curve shown beside with ID label: 21866
ALL:SELEC.

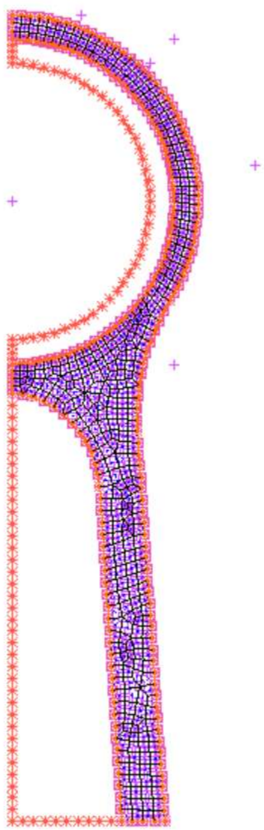
Mesh generation

Connecting rod 2D model

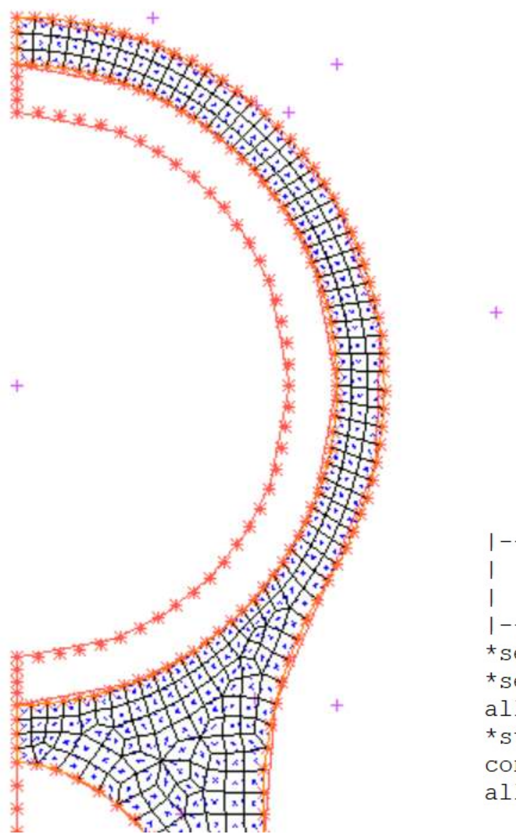


MAIN
MESH GENERATION
AUTOMESH
2-D PLANAR MESHING
QUADRILATERAL (ADV FRNT)
QUAD MESH!
Select manually the curves shown beside with ID label:
21854
12618
21849
21871
21837
12712
21863
21869
21818
21862
21866
 # | End of List
ALL:SELEC.

Mesh generation
Connecting rod 2D model



Global view



Detailed view

The mesh is uniformally distributed both in the circumferential (equispaced) and in radial direction (4 divisions).

Finally, store these elements in a collector named `conrod_shank.`

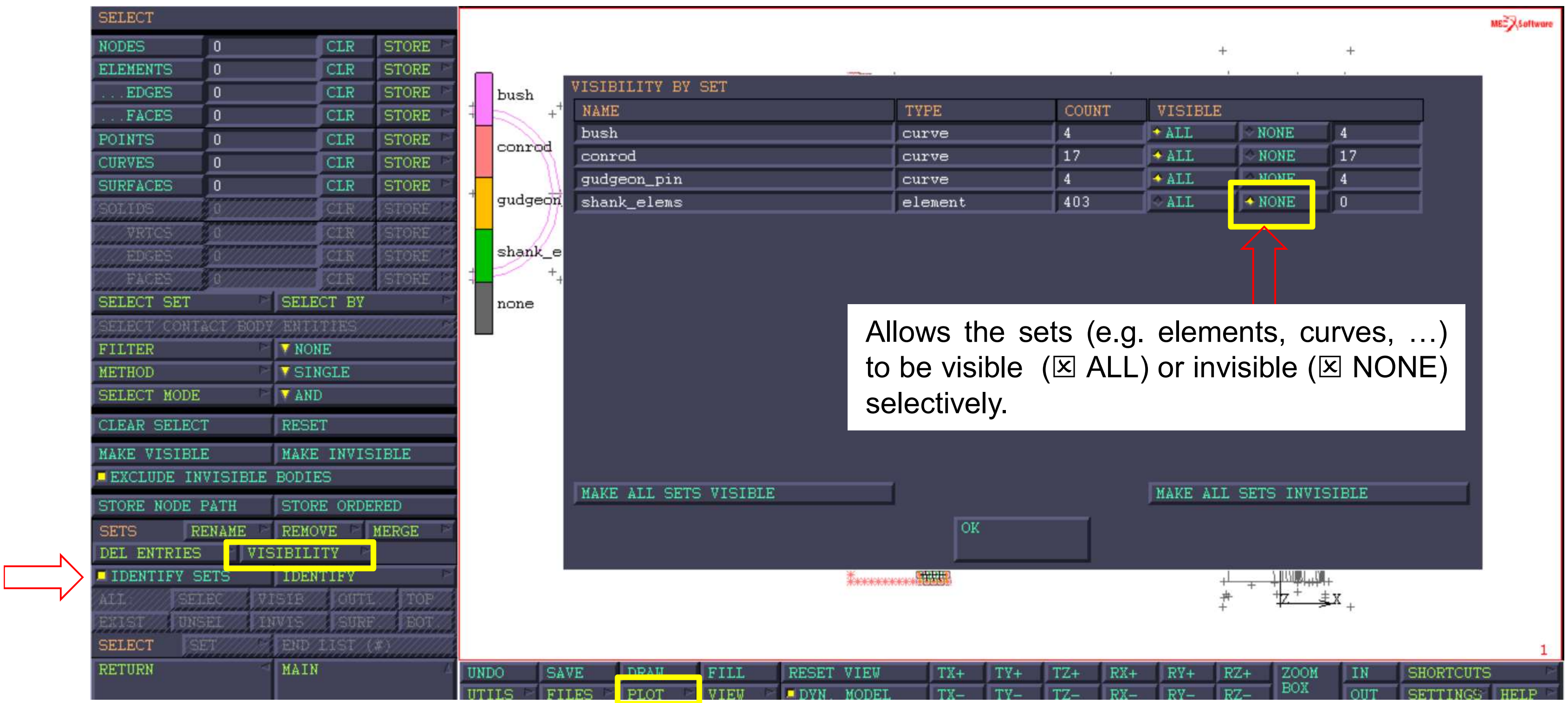
```
|-----------------------------------
|   STORE and PLOT the elements in a collector
|   named shank_elems
|-----------------------------------
*select_clear_elements
*select_elements
all_existing
*store_elements
conrod_shank
all_selected
```

Mesh generation

Connecting rod 2D model

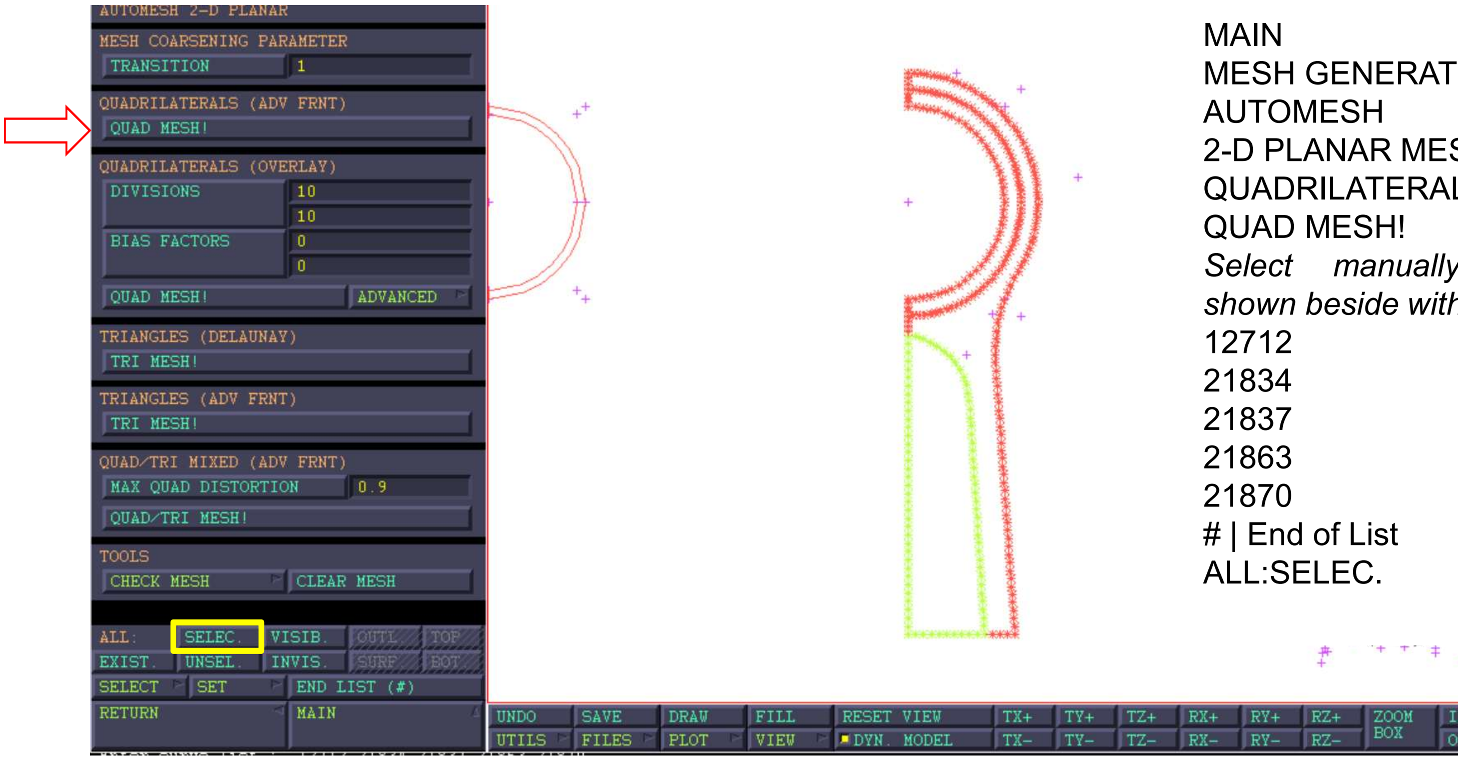
To identify sets:

PLOT
SELECT
☒ IDENTIFY SETS
VISIBILITY
`SHANK_ELEMS` ☒ NONE
OK



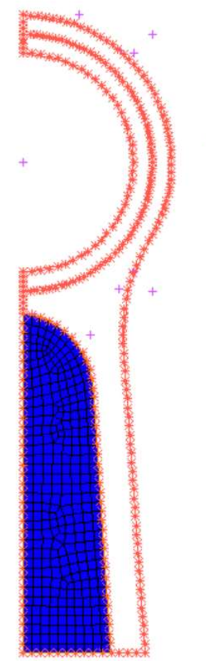
Mesh generation

Connecting rod 2D model

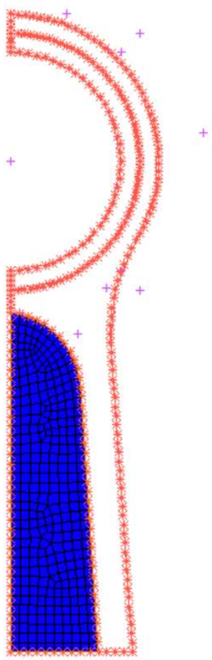


MAIN
MESH GENERATION
AUTOMESH
2-D PLANAR MESHING
QUADRILATERAL (ADV FRNT)
QUAD MESH!
Select manually the curves shown beside with ID label:
12712
21834
21837
21863
21870
| End of List
ALL:SELEC.

The pocket



Mesh generation
Connecting rod 2D model

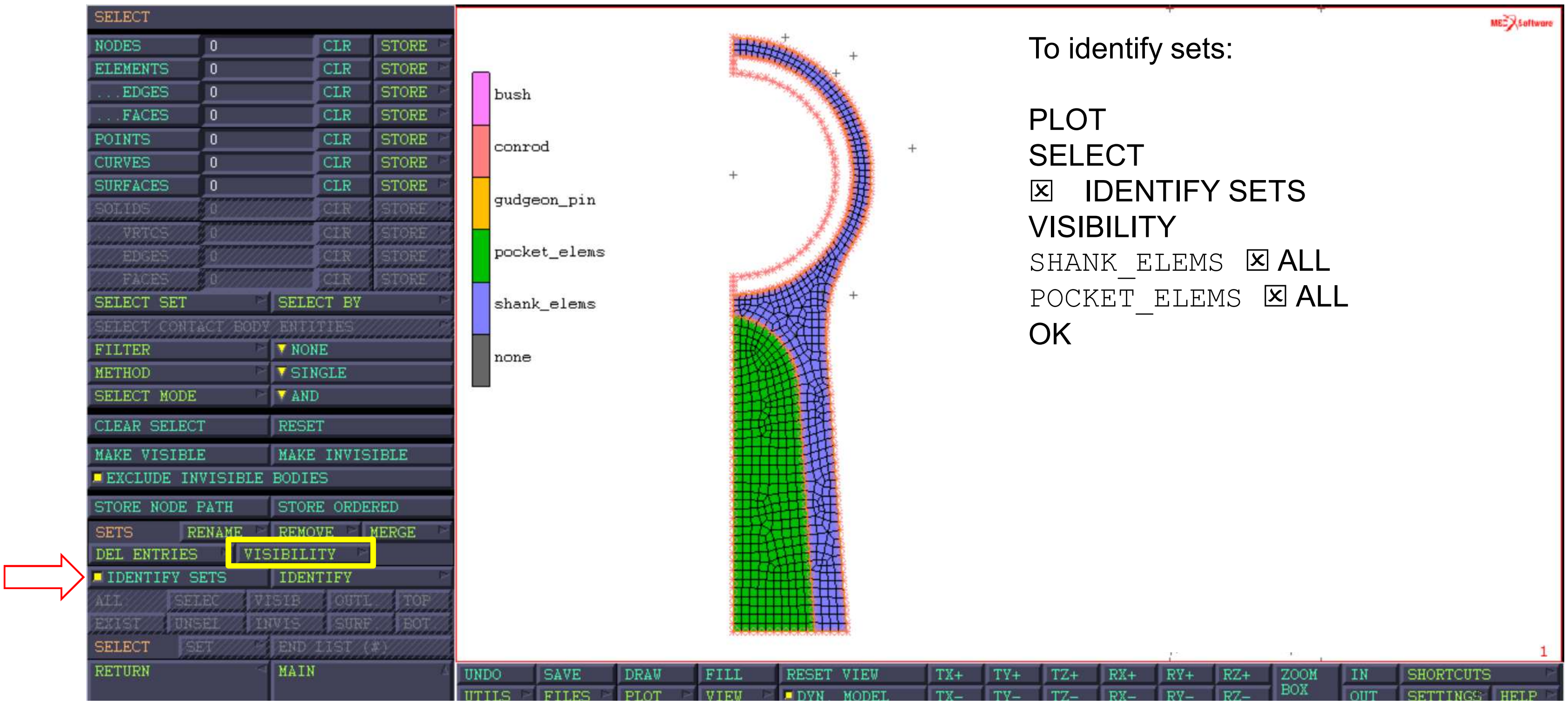


```
|-----------------------------------
|   STORE and PLOT the elements in a collector
|   named shank_elems
|-----------------------------------
*select_clear_elements
*select_elements
all_visible
*store_elements pocket_elems
all_selected
```

The **visible** elements alone must be stored named as `pocket_elemts.`

Mesh generation

Connecting rod 2D model



Mesh generation
Elements selection

Element 3 — **Plane Stress Quadrilateral**

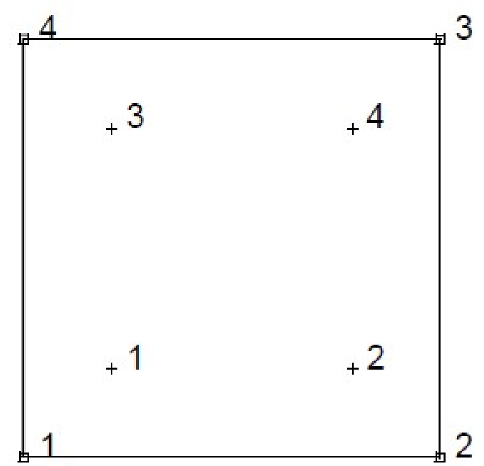
Element 3 is a four-node, isoparametric, arbitrary quadrilateral written for plane stress applications. As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior. The shear (or bending) characteristics can be improved by using alternative interpolation functions. This assumed strain procedure is flagged through the GEOMETRY option.

In general, one needs more of these lower-order elements than the higher-order elements such as 26 or 53. Hence, use a fine mesh.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using four-point Gaussian integration.

All constitutive models can be used with this element.



Mesh generation
Elements selection

Element 26 — **Plane Stress, Eight-node Distorted Quadrilateral**

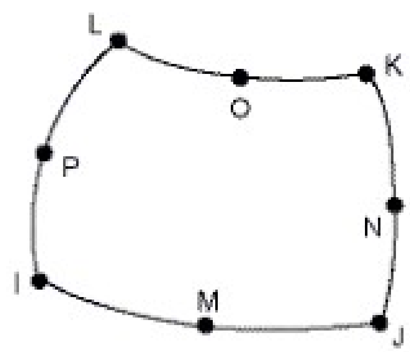
Element type 26 is an eight-node, isoparametric, arbitrary quadrilateral written for plane stress applications. This element uses biquadratic interpolation functions to represent the coordinates and displacements. This allows for a more accurate representation of the strain fields in elastic analyses than lower order elements.

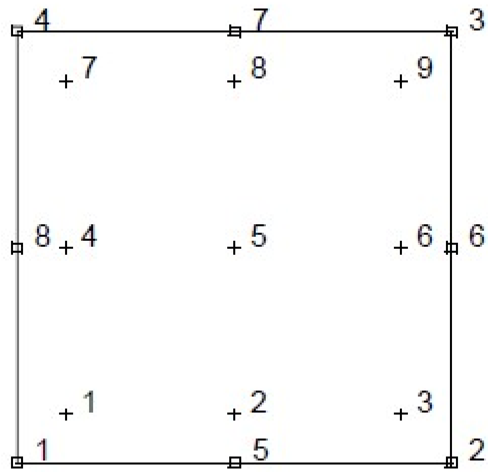
Lower-order elements, such as type 3, are preferred in contact analyses.

The stiffness of this element is formed using eight-point Gaussian integration.

All constitutive models can be used with this element.

With extra nodes at the midsides, we can assume quadratic variations in element. This is rather like fitting a given curve with a series of parabolas, instead of straight lines. This makes it easier to fit circles, …



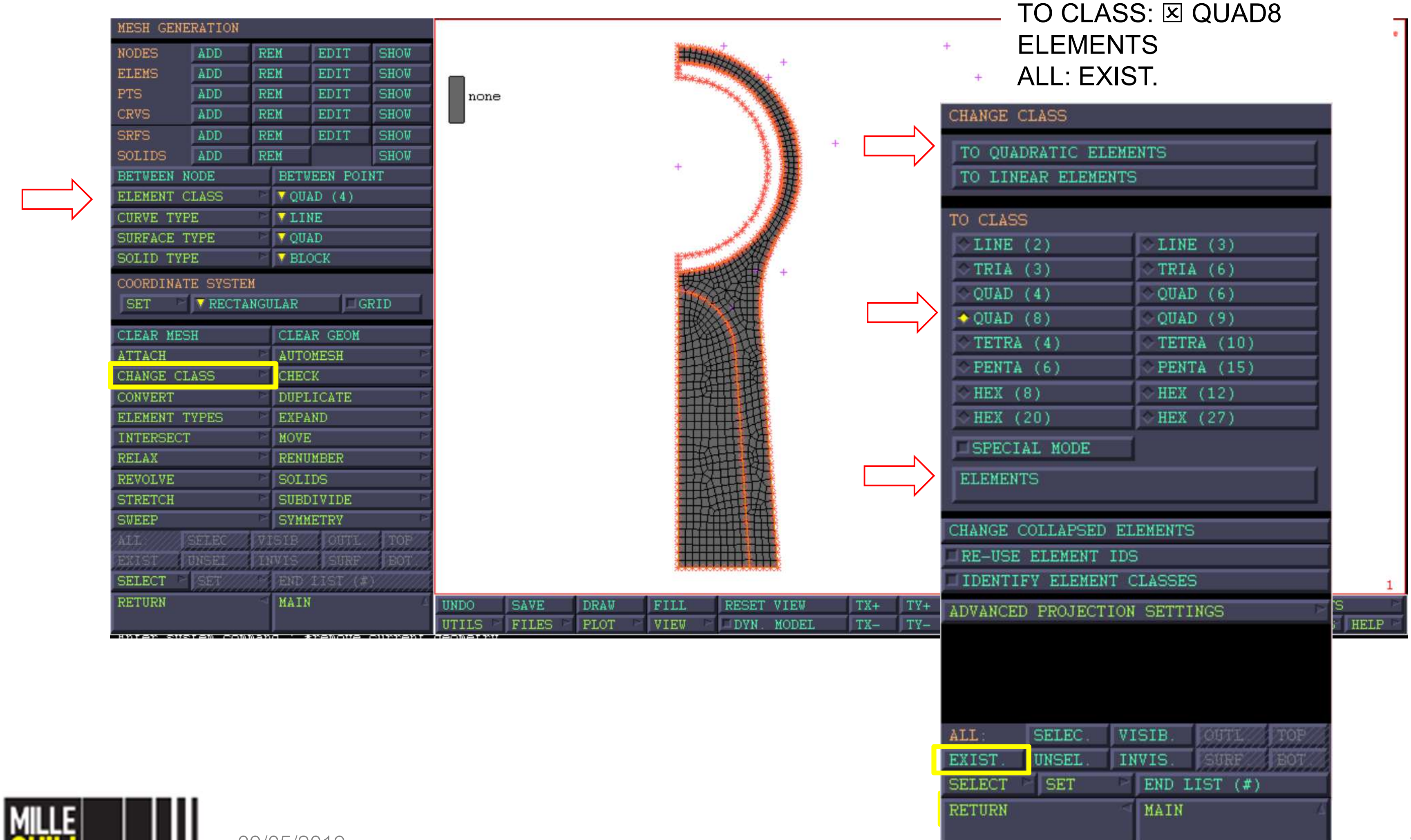


Mesh generation

Elements selection

To change the Elements class:

MAIN
MESH GENERATION
CHANGE CLASS
TO QUADRATIC ELEMENTS
TO CLASS: ☒ QUAD8
ELEMENTS
ALL: EXIST.



Mesh generation

Connecting rod 2D model – small end mesh

```
|---------------------------------
|   CREATE NODES:
|  -  AT THE SMALL END INNER RADIUS(ID 2459)
|  -  AT MIDDLE SMALL END  (ID 2460)
|  -  AT OUTER SMALL END (ID 48)
|---------------------------------
*add_nodes
0
23/2
0
*add_nodes
0
25.003/2
0
|---------------------------------
|   CREATE NODE AT THE SMALL END INNER RADIUS
|   create a 1D quadratic element (line3)
|   1 node: inner node (ID 2459)
|   2 node: outer node (ID 48)
|   3 node: middle node (ID 2460)
|---------------------------------
*set_element_class line3
*add_elements
 2459
 48
 2460
```

The small end connecting rod mesh will be performed by the expansion of 1D quadratic element (line3) to obtained quadratic and planar elements. This elements are characterized by 8 nodes and 9 integration points.

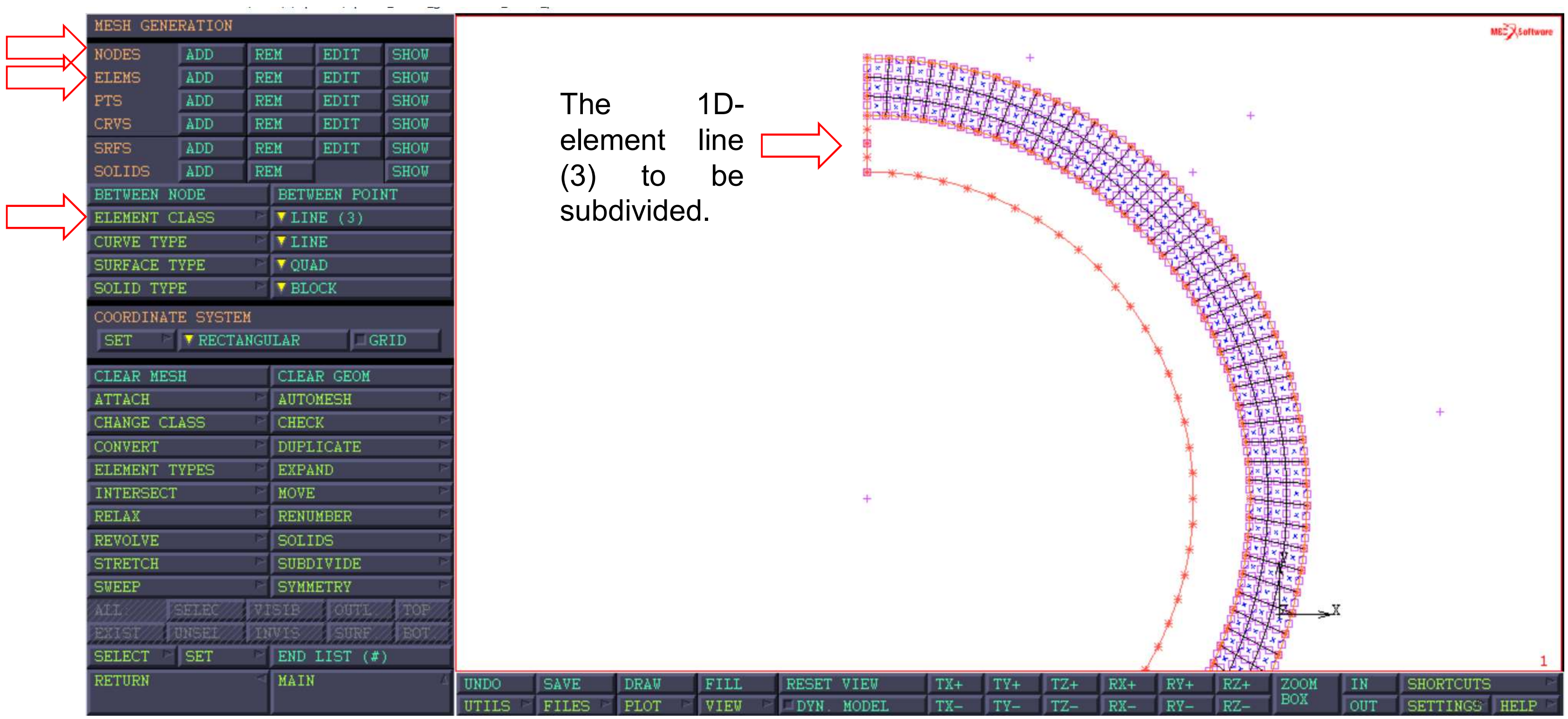
To create a 1D line(3) element the selection of the nodes is prescribed as follows:

- The first and the second nodes must be the outer nodes of the element;
- The third node is the node located in between of the further nodes.

MILLE CHILI

Mesh generation

Connecting rod 2D model – small end mesh



Mesh generation

Connecting rod 2D model – small end mesh

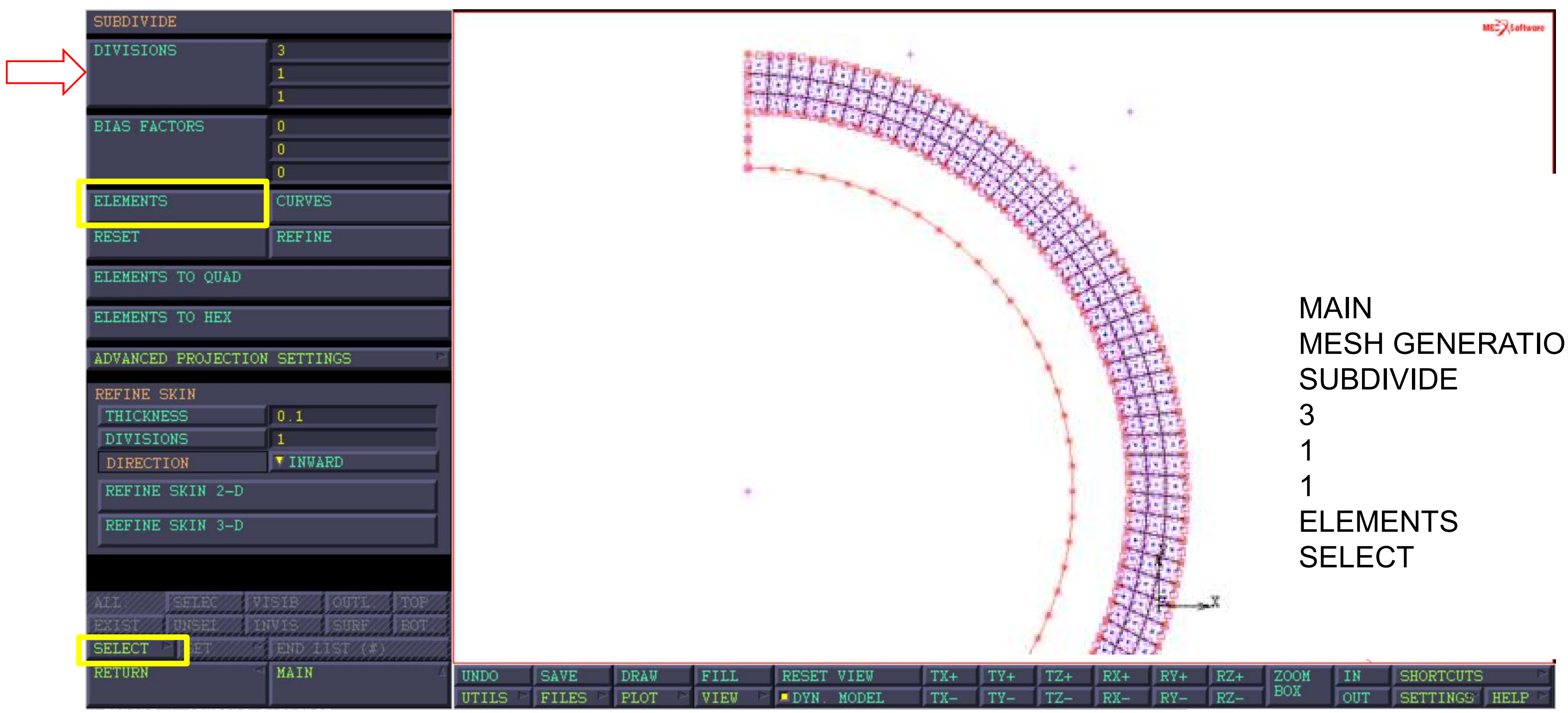
```
|----------------------------------
|   MESH GENERATION: subdivide
|   The line3 element will be subdivided in three.
|----------------------------------
*subdivide_reset
*sub_divisions
3
1
1
*subdivide_elements
all_selected
|----------------------------------
|   select the elements previously subdivided by class (line3)
|----------------------------------
*select_clear_elements
*select_elements
*select_elements_class line3
```

This 1D quadratic element (line3) has 2.003 mm length. To obtain a fine mesh of the conrod small end, a subdivision of this 1D element is required.

Three divisions will be assessed.

Mesh generation

Connecting rod 2D model – small end mesh

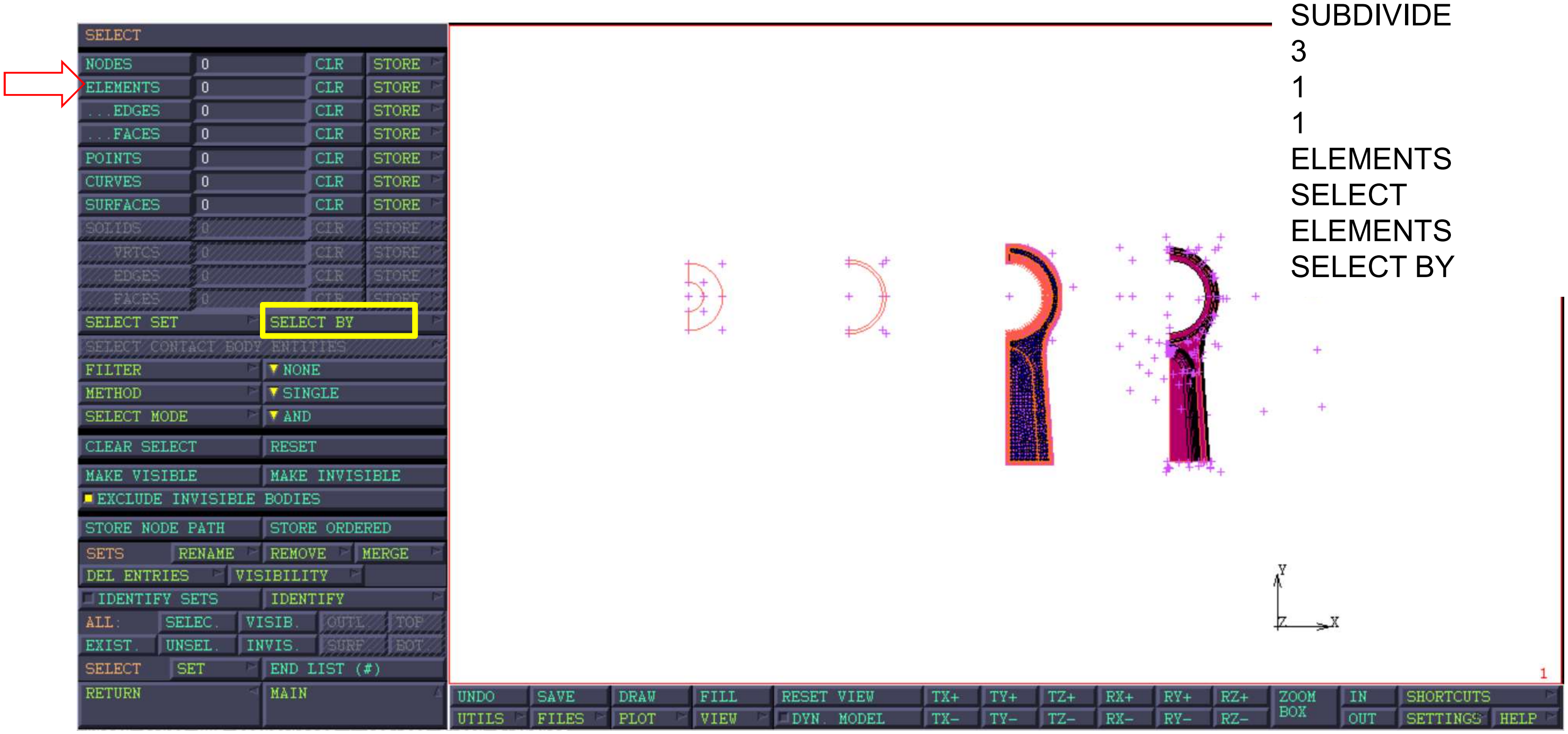


MAIN
MESH GENERATION
SUBDIVIDE
3
1
1
ELEMENTS
SELECT

Mesh generation

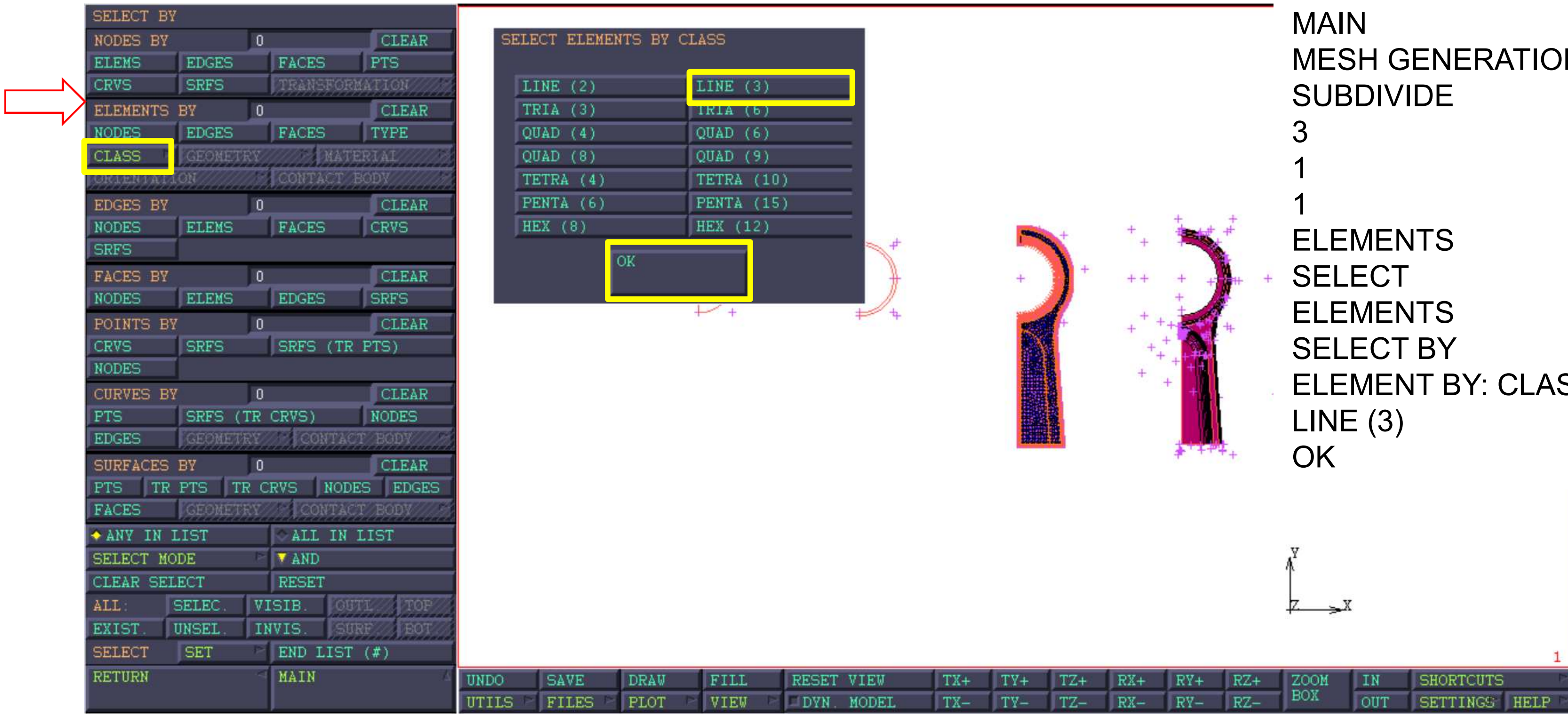
Connecting rod 2D model – small end mesh

MAIN
MESH GENERATION
SUBDIVIDE
3
1
1
ELEMENTS
SELECT
ELEMENTS
SELECT BY



Mesh generation

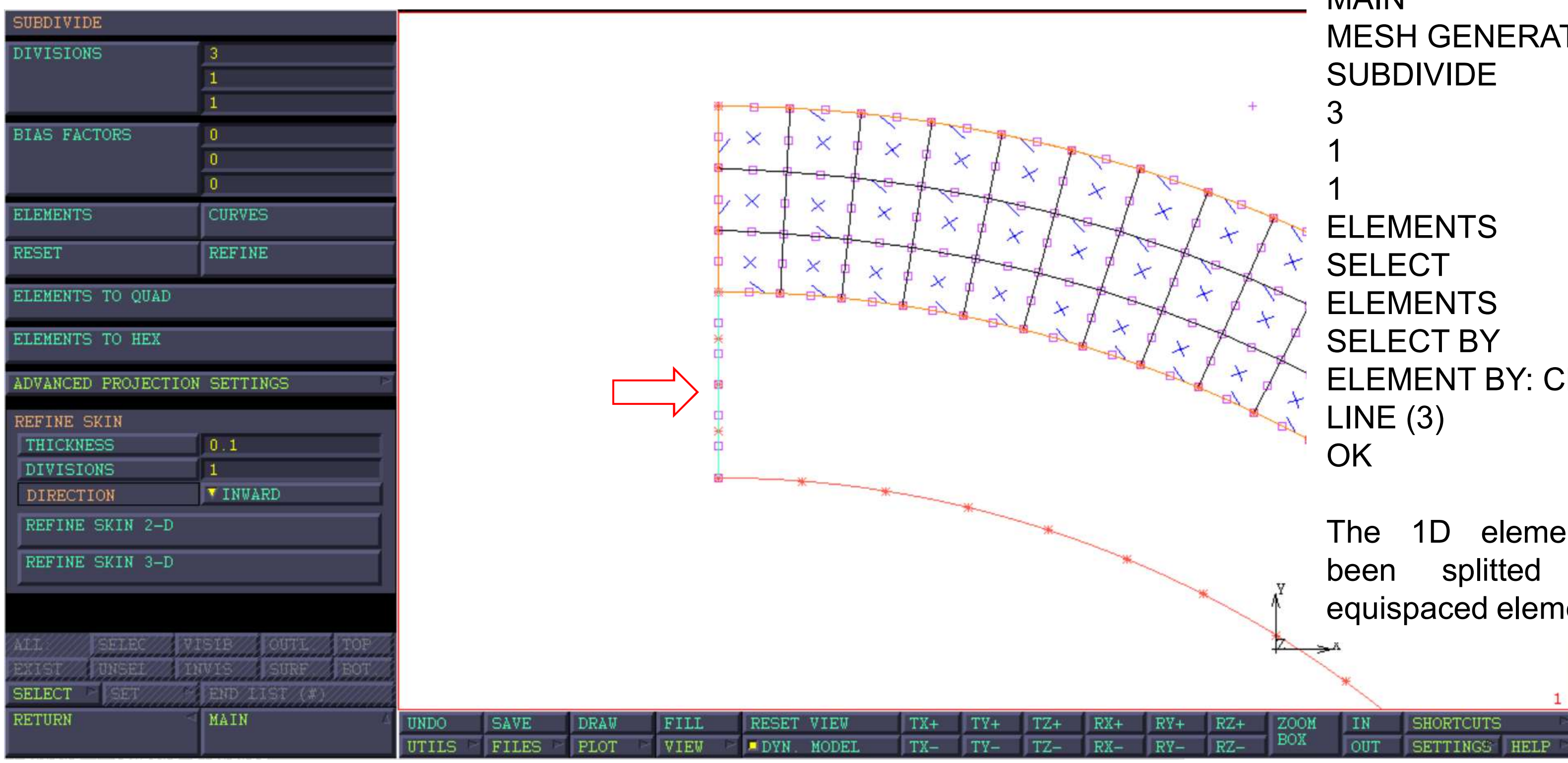
Connecting rod 2D model – small end mesh



MAIN
MESH GENERATION
SUBDIVIDE
3
1
1
ELEMENTS
SELECT
ELEMENTS
SELECT BY
ELEMENT BY: CLASS
LINE (3)
OK

Mesh generation

Connecting rod 2D model – small end mesh

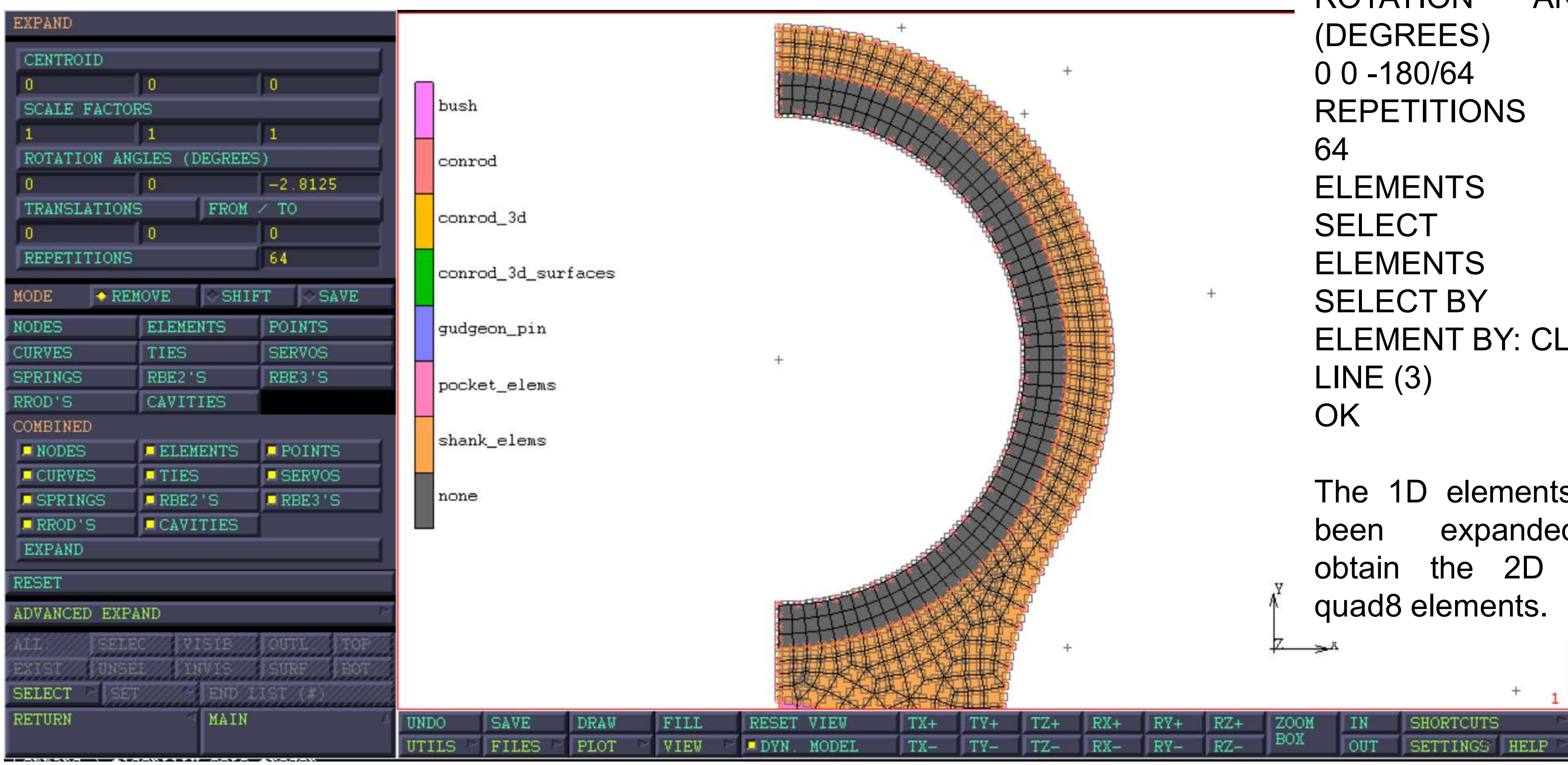


MAIN
MESH GENERATION
SUBDIVIDE
3
1
1
ELEMENTS
SELECT
ELEMENTS
SELECT BY
ELEMENT BY: CLASS
LINE (3)
OK

The 1D element has been splitted in 3 equispaced elements.

Mesh generation

Connecting rod 2D model – small end mesh



MAIN
MESH GENERATION
EXPAND
CENTROID
0 0 0
ROTATION ANGLES (DEGREES)
0 0 -180/64
REPETITIONS
64
ELEMENTS
SELECT
ELEMENTS
SELECT BY
ELEMENT BY: CLASS
LINE (3)
OK

The 1D elements have been expanded to obtain the 2D planar quad8 elements.

Mesh generation
Connecting rod 2D model – small end mesh

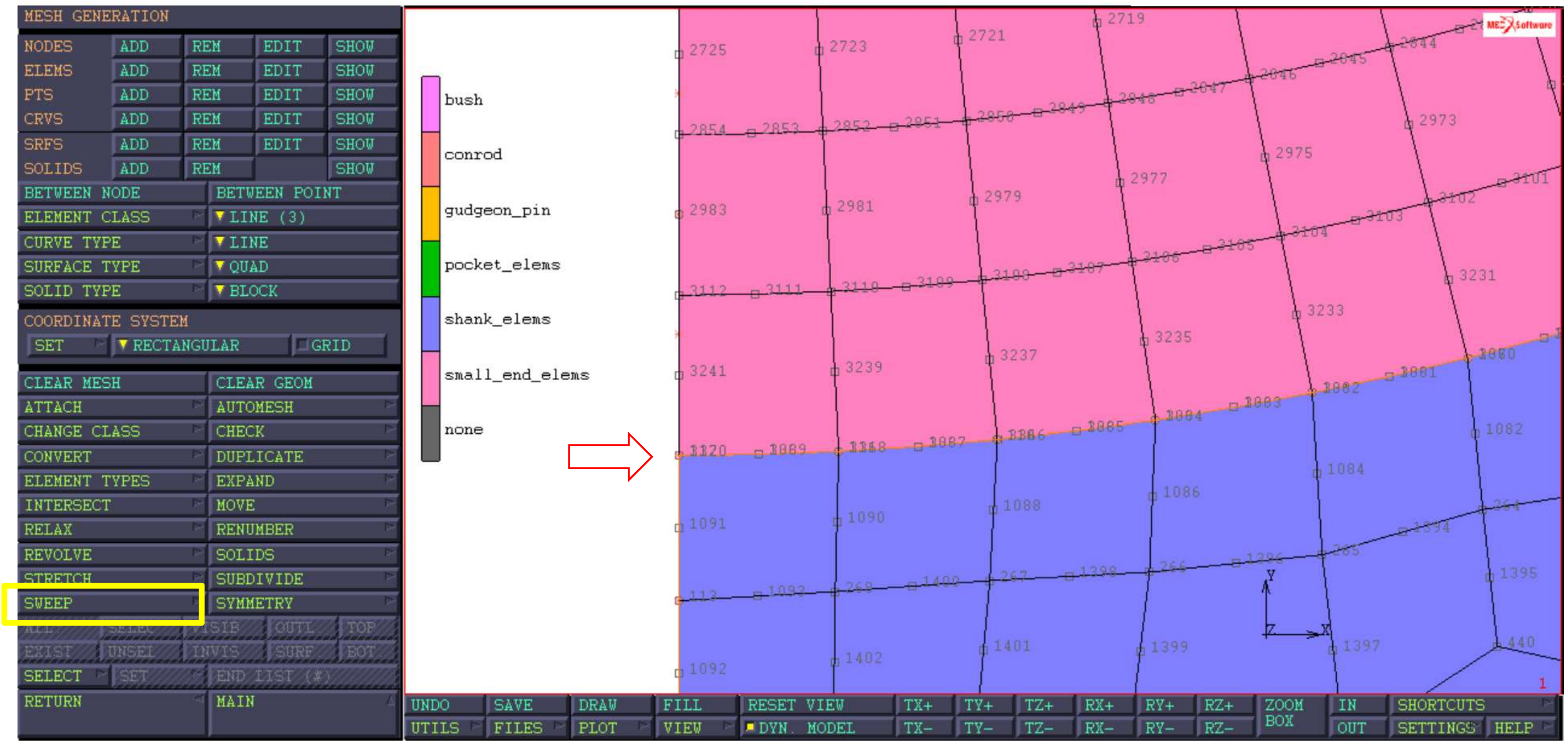
```
|-----------------------------------
*visible_all_sets
*identify_sets *regen
*invisible_set shank_elems
*invisible_set pocket_elems
*select_clear_elements
*select_elements
all_visible
*store_elements small_end_elems
all_selected
*select_clear_elements
*visible_all_sets
|-----------------------------------
```

At the end, these elements are stored, and the set is called `small_end_elems`

Mesh generation
Sweep

The conrod regions must be sweeped, its borders are characterized by superposed nodes

MAIN
MESH GENERATION
SWEEP



Mesh generation

Sweep

The conrod regions must be sweeped, its borders are characterized by superposed nodes

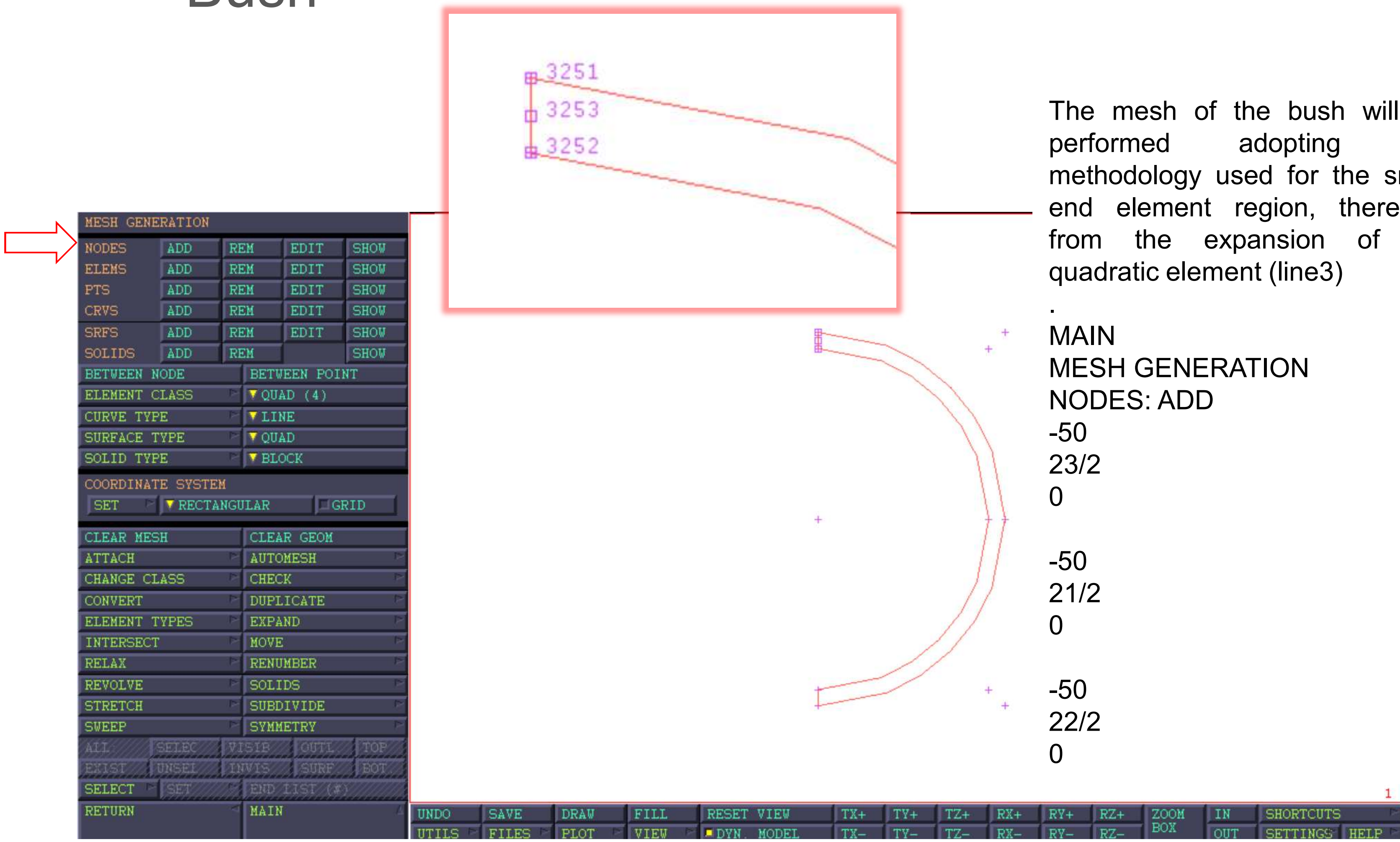
MAIN
MESH GENERATION
SWEEP
Tolerance
0.05
NODES
ALL: EXISTS.

In the command prompt, you read:

```
Delecting 219 duplicate nodes!
Delecting 0 collapsed elements!
```

Mesh generation

Bush



The mesh of the bush will be performed adopting the methodology used for the small end element region, therefore from the expansion of 1D quadratic element (line3)

.

MAIN
MESH GENERATION
NODES: ADD
-50
23/2
0

-50
21/2
0

-50
22/2
0

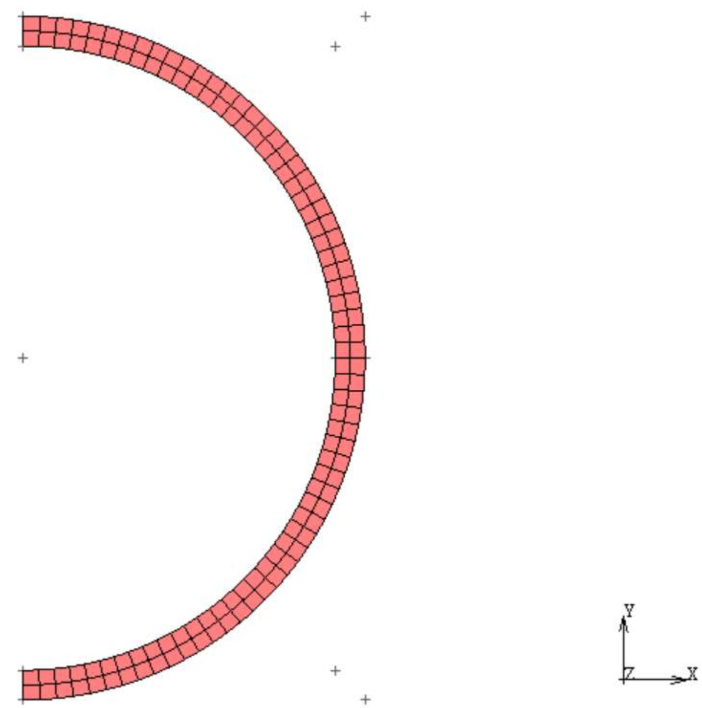
Mesh generation

Bush

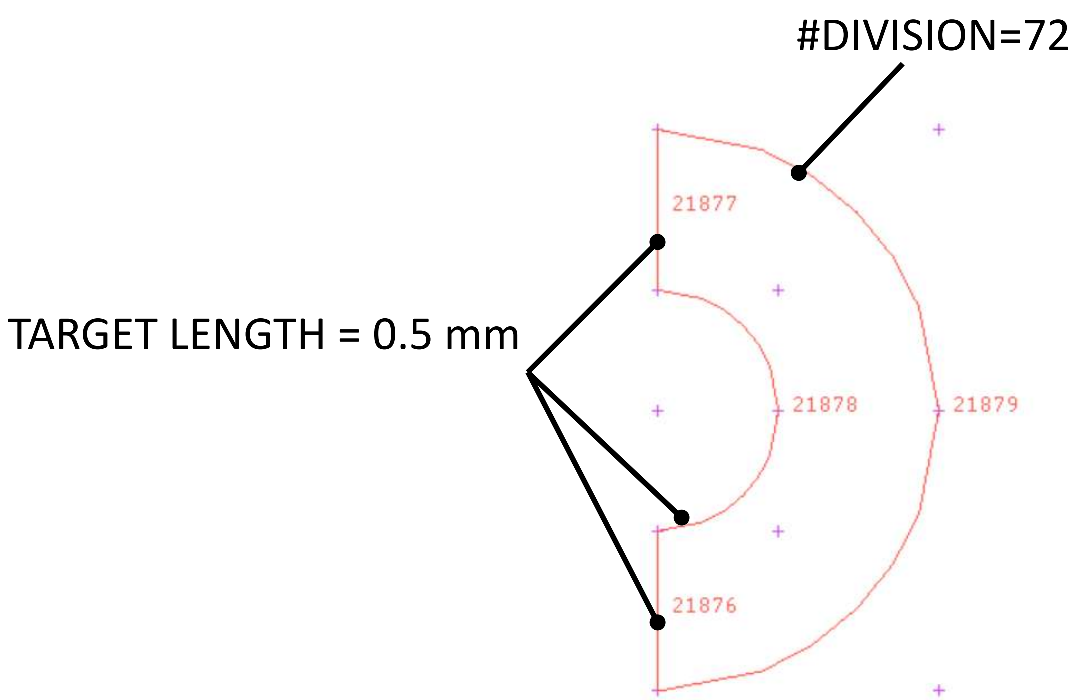
```
|---------------------------------
|   create a 1D quadratic element (line3)
|   1 node: outer node (ID 3251)
|   2 node: inner node (ID 3252)
|   3 node: middle node (ID 3253)
|---------------------------------
*set_element_class line3
*add_elements
 3251
 3252
 3253
|---------------------------------
|   select the element to be subdivided by class (line3)
|---------------------------------
*select_clear_elements
*select_elements
*select_elements_class line3
|---------------------------------
|   MESH GENERATION: subdivide
|   The line3 element will be subdivided in two.
|---------------------------------
*subdivide_reset
*sub_divisions
2
1
1
*subdivide_elements
all_selected
|---------------------------------
|   select the elements previously subdivided by class (line3)
|---------------------------------
*select_clear_elements
*select_elements
*select_elements_class line3
|---------------------------------
```

Mesh generation
Bush

```
|----------------------------------
|   MESH GENERATION: expand
|   From 1D quadratic elements (line3)
|   to 2D-planar quadratic elements (quad8).
|----------------------------------
*expand_reset
*set_expand_point
-50
0
0
*set_expand_repetitions 64
*set_expand_rotation z -180/64
*expand_elements
all_selected
|----------------------------------
|   STORE and PLOT the elements in a collector
|   named bush_elems
|----------------------------------
*visible_all_sets
*identify_sets *regen
*invisible_set shank_elems
*invisible_set pocket_elems
*invisible_set small_end_elems
*select_clear_elements
*select_elements
all_visible
*store_elements bush_elems
all_selected
*select_clear_elements
*visible_all_sets
|----------------------------------
```



Mesh generation
Gudgeon pin



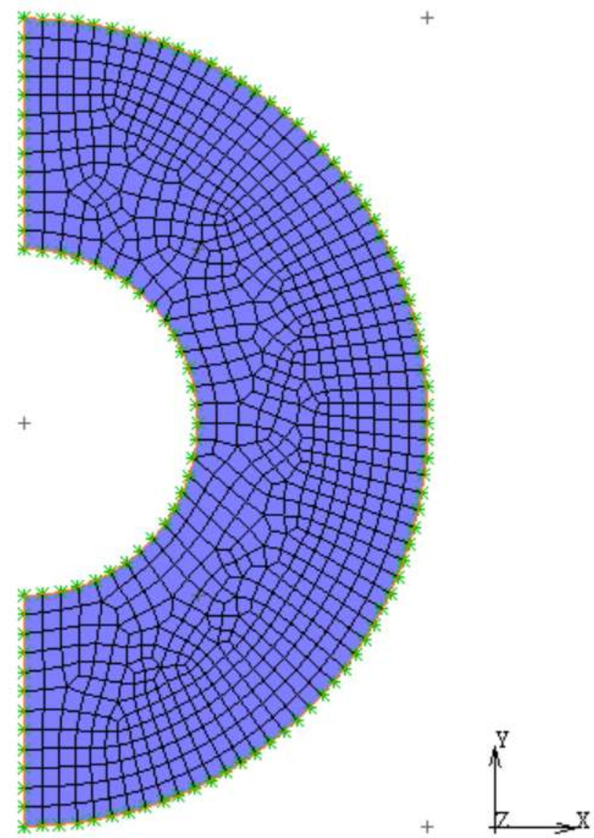
The mesh of gudgeon pin will be performed adopting the methodology used for the pocket conrod element region, therefore subdividing its curves with a prescribed number of divisions or with a prescribed target length of the elements.

```
|-----------------------------------
| Outer gudgeon pin curve
|-----------------------------------
*select_clear_curves
*set_curve_div_type_fix_ndiv
*set_curve_div_num
72
*set_curve_div_rest_off
*apply_curve_divisions
 21879
# | End of List
*select_clear_curves
|-----------------------------------
| Inner and vertical gudgeon pin curves
|-----------------------------------
*set_curve_div_type_fix
*set_curve_div_type_fix_avgl
*set_curve_div_avgl
0.5
*set_curve_div_rest_evn
*set_curve_div_applyrest_cvs
*apply_curve_divisions
21876
21877
21878
all_selected
|-----------------------------------
```

Mesh generation

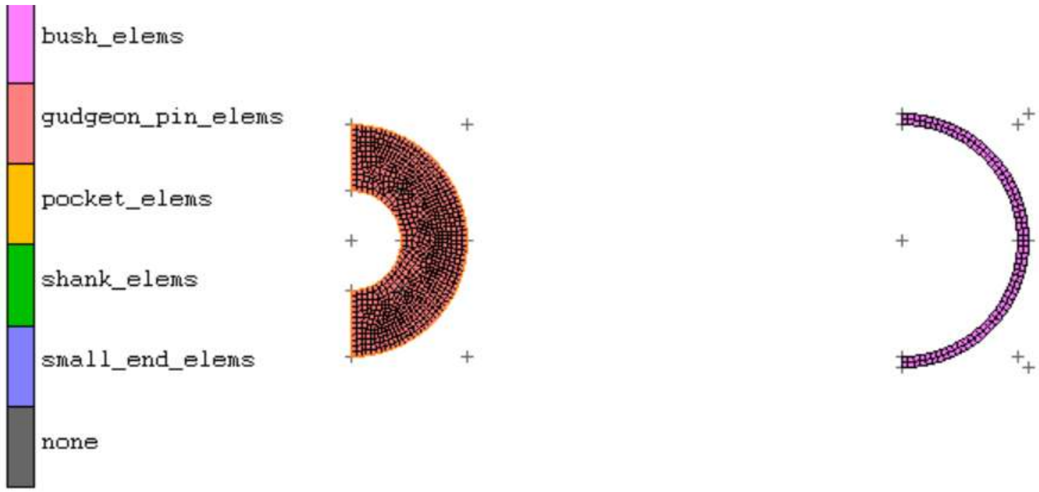
Gudgeon pin

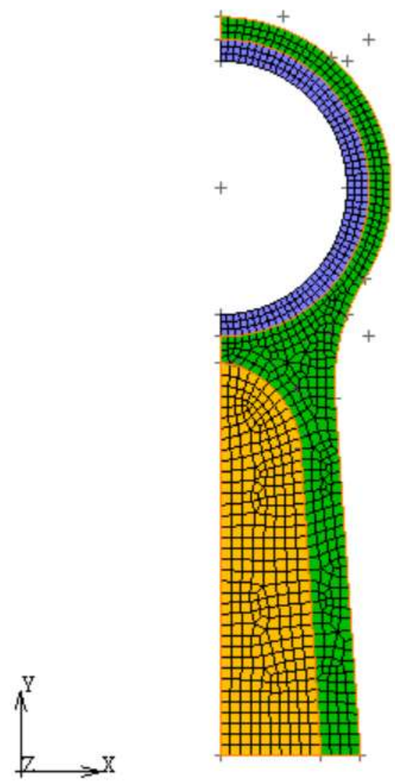
```
|----------------------------------
|   AUTOMESH: Curve divisions
|   planar mesh Quadrilateral 4-nodes elements
|   gudgeon pin
|----------------------------------
*af_planar_quadmesh
gudgeon_pin
*select_clear_curves
|----------------------------------
|   STORE and PLOT the elements in a collector
|   named gudgeon_pin_elems
|----------------------------------
*invisible_set bush_elems
*invisible_set pocket_elems
*invisible_set shank_elems
*invisible_set small_end_elems
*select_elements
all_visible
*store_elements gudgeon_pin_elems
all_selected
*select_clear_elements
*visible_all_sets
|----------------------------------
|   FROM QUAD4 TO QUAD8
|   CHANGE ELEMENTS CLASS
|----------------------------------
*change_elements_class
*change_elements_quadratic
*set_change_class quad8
*change_elements_class
all_visible
|----------------------------------
```



Mesh generation

The components sets

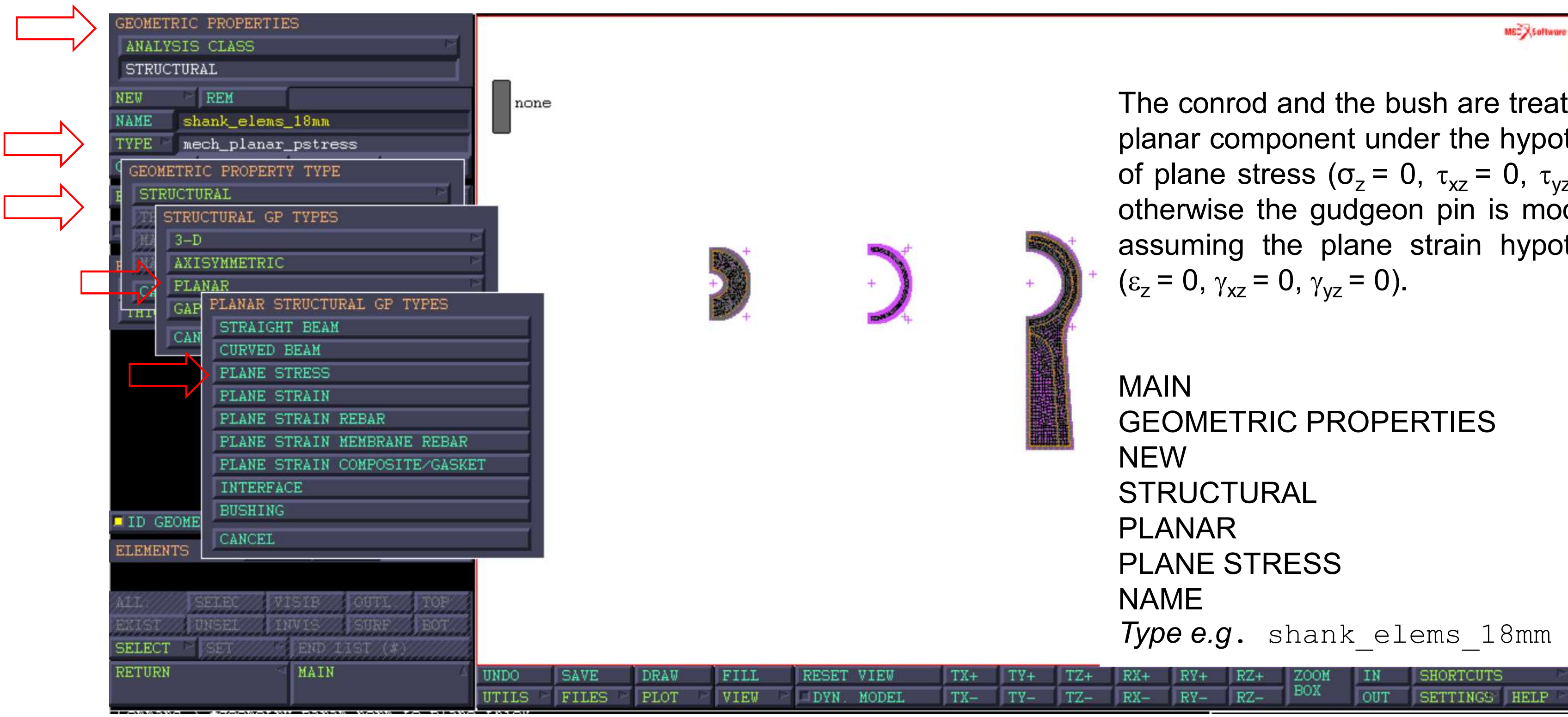




Component	Geometric properties	Thickness [mm]
Shank_elems	Plane stress	18
Pocket_elems	Plane stress	4
Small_end_elems	Plane stress	20
Bush_elems	Plane stress	20
Gudgeon_pin_elems	Plane strain	60

Mesh generation

The geometric properties

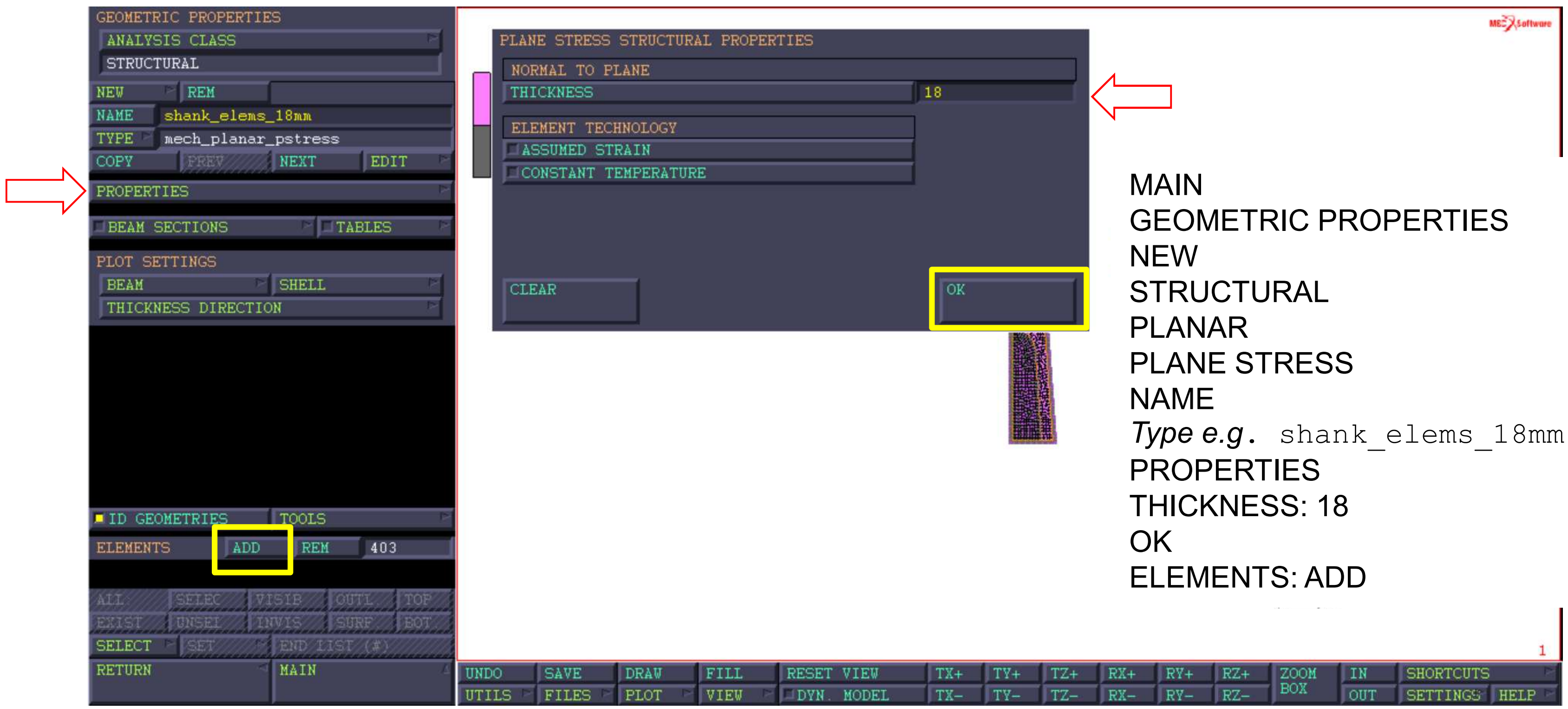


The conrod and the bush are treated as planar component under the hypothesis of plane stress ($\sigma_z = 0$, $\tau_{xz} = 0$, $\tau_{yz} = 0$); otherwise the gudgeon pin is modelled assuming the plane strain hypothesis ($\varepsilon_z = 0$, $\gamma_{xz} = 0$, $\gamma_{yz} = 0$).

MAIN
GEOMETRIC PROPERTIES
NEW
STRUCTURAL
PLANAR
PLANE STRESS
NAME
Type e.g. `shank_elems_18mm`

Mesh generation

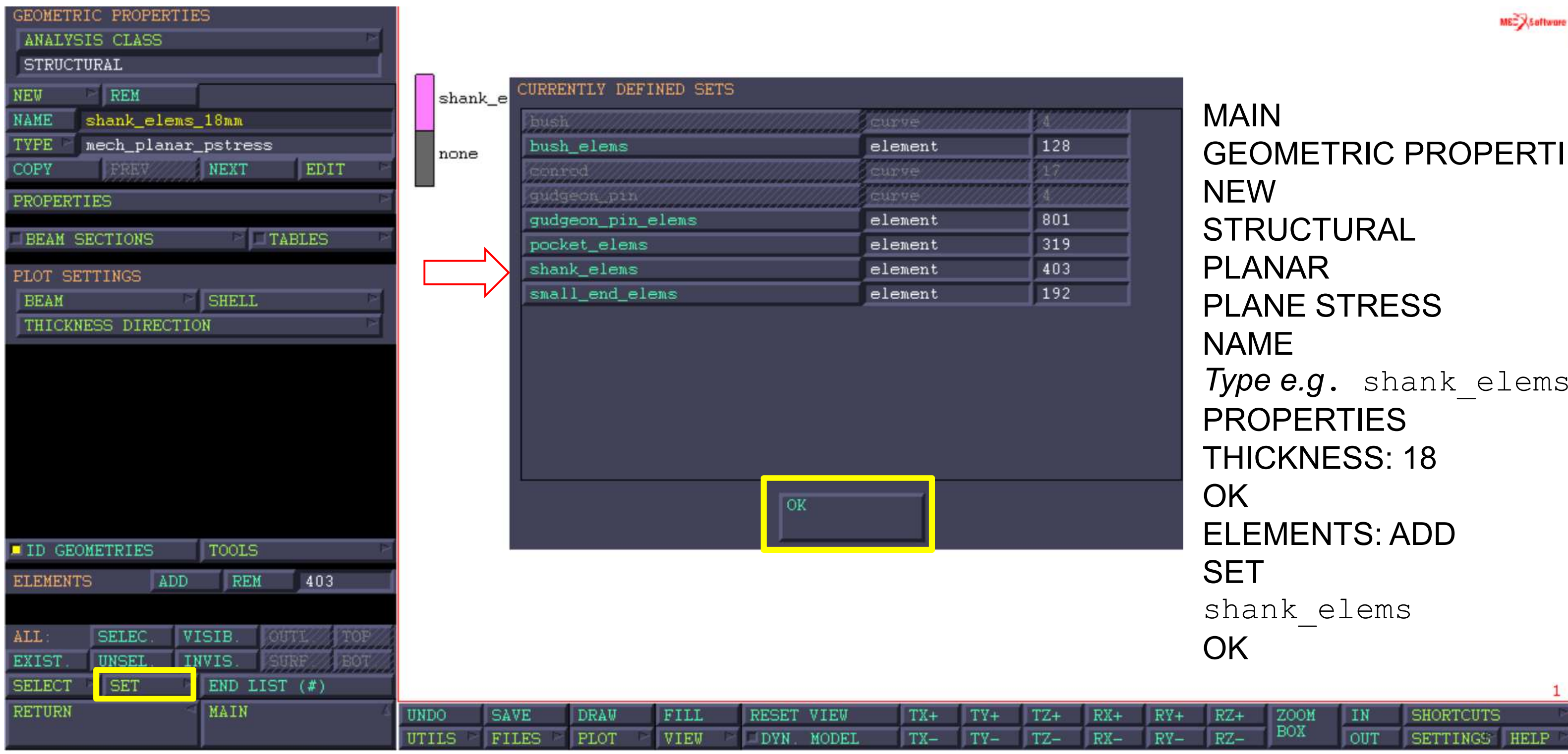
The geometric properties



MAIN
GEOMETRIC PROPERTIES
NEW
STRUCTURAL
PLANAR
PLANE STRESS
NAME
Type e.g. `shank_elems_18mm`
PROPERTIES
THICKNESS: 18
OK
ELEMENTS: ADD

Mesh generation

The geometric properties



MAIN
GEOMETRIC PROPERTIES
NEW
STRUCTURAL
PLANAR
PLANE STRESS
NAME
Type e.g. `shank_elems_18mm`
PROPERTIES
THICKNESS: 18
OK
ELEMENTS: ADD
SET
`shank_elems`
OK

Mesh generation

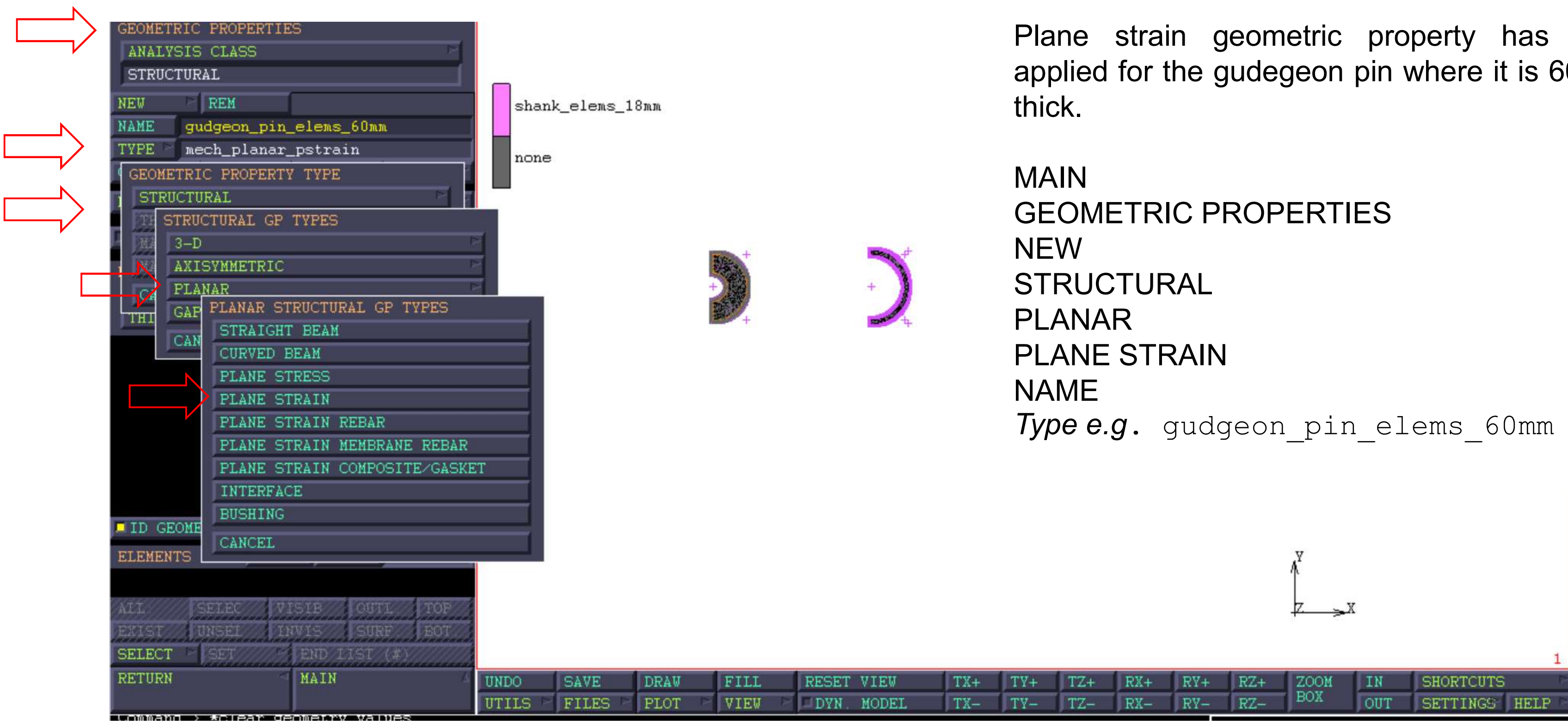
Geometric properties

```
|----------------------------------
|   POCKET ELEMS GEOMETRIC PROPERTIES
|----------------------------------
*new_geometry
*geometry_type mech_planar_pstress
*geometry_name pocket_elems_4mm
*geometry_param norm_to_plane_thick
4
*add_geometry_elements
pocket_elems
|----------------------------------
|   SMALL END ELEMS GEOMETRIC PROPERTIES
|----------------------------------
*new_geometry
*geometry_type mech_planar_pstress
*geometry_name small_end_elems_20mm
*geometry_param norm_to_plane_thick
20
*add_geometry_elements
small_end_elems
|----------------------------------
|   BUSH ELEMS GEOMETRIC PROPERTIES
|----------------------------------
*new_geometry
*geometry_type mech_planar_pstress
*geometry_name bush
*geometry_name bush_elems_20mm
*geometry_param norm_to_plane_thick
20
*add_geometry_elements
bush_elems
|----------------------------------
```

The plane stress geometric properties have been applied to the further components, setting propertly the thickness as shown beside.

Mesh generation

Geometric properties

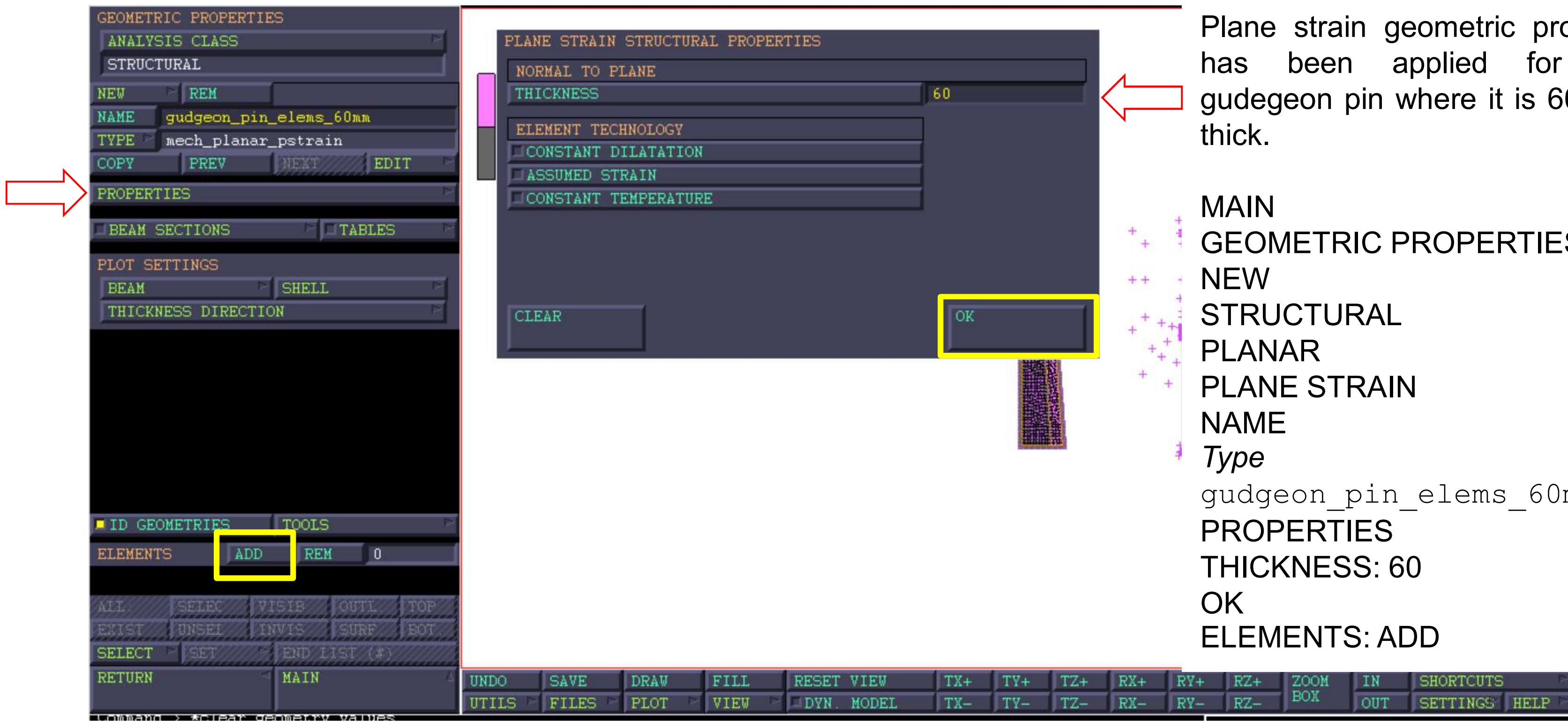


Plane strain geometric property has been applied for the gudegeon pin where it is 60 mm thick.

MAIN
GEOMETRIC PROPERTIES
NEW
STRUCTURAL
PLANAR
PLANE STRAIN
NAME
Type e.g. `gudgeon_pin_elems_60mm`

Mesh generation

Geometric properties

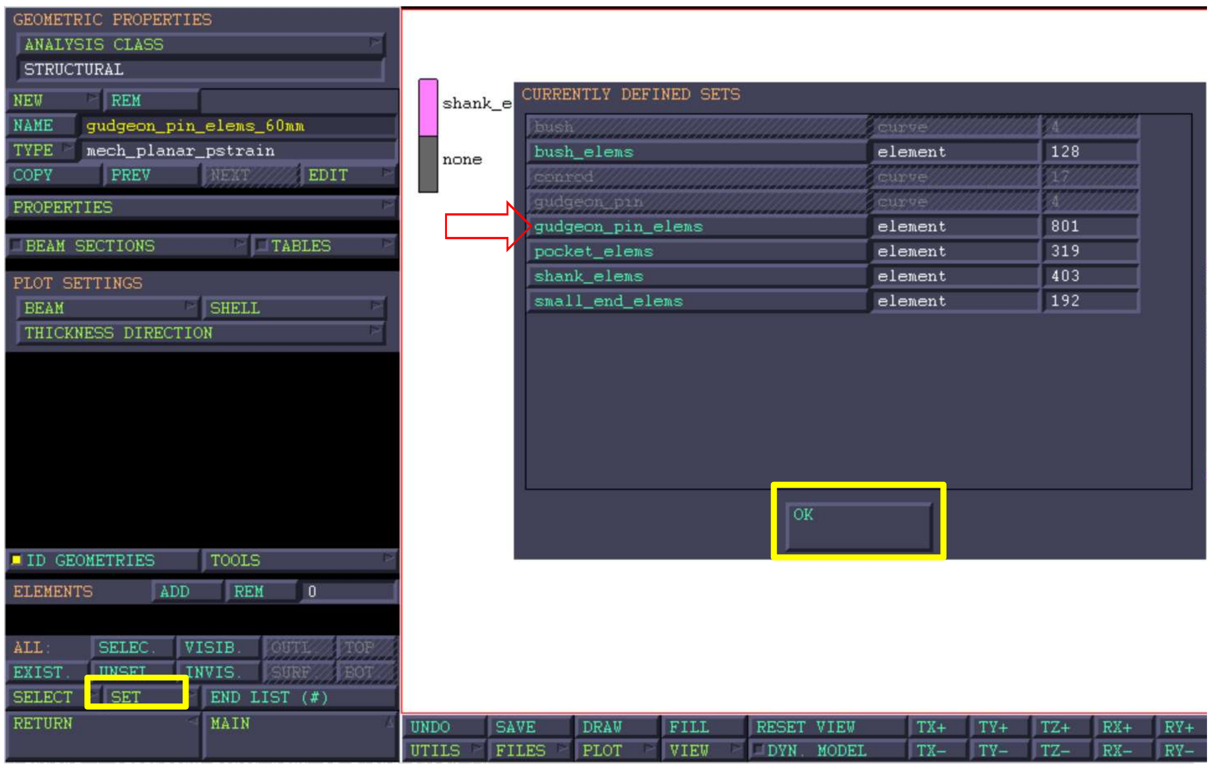


Plane strain geometric property has been applied for the gudegeon pin where it is 60 mm thick.

MAIN
GEOMETRIC PROPERTIES
NEW
STRUCTURAL
PLANAR
PLANE STRAIN
NAME
Type *e.g.* `gudgeon_pin_elems_60mm`
PROPERTIES
THICKNESS: 60
OK
ELEMENTS: ADD

Mesh generation

Geometric properties



Plane strain geometric property has been applied for the gudegeon pin where it is 60 mm thick.

MAIN
GEOMETRIC PROPERTIES
NEW
STRUCTURAL
PLANAR
PLANE STRAIN
NAME
Type *e.g.* `gudgeon_pin_elems_60mm`
PROPERTIES
THICKNESS: 60
OK
ELEMENTS: ADD
SET
`gudgeon_pin_elems`
OK

Mesh generation

Material properties

```
|-----------------------------------
|   START MATERIALS
|-----------------------------------
*new_mater standard
*mater_option general:state:solid
*mater_name
steel
*mater_option structural:type:elast_plast_iso
*mater_param structural:youngs_modulus
210000
*mater_param structural:poissons_ratio
0.3
*add_mater_elements
all_existing
*identify_materials *regen
*identify_none *regen
|-----------------------------------
|   FINISH MATERIALS
|-----------------------------------
```

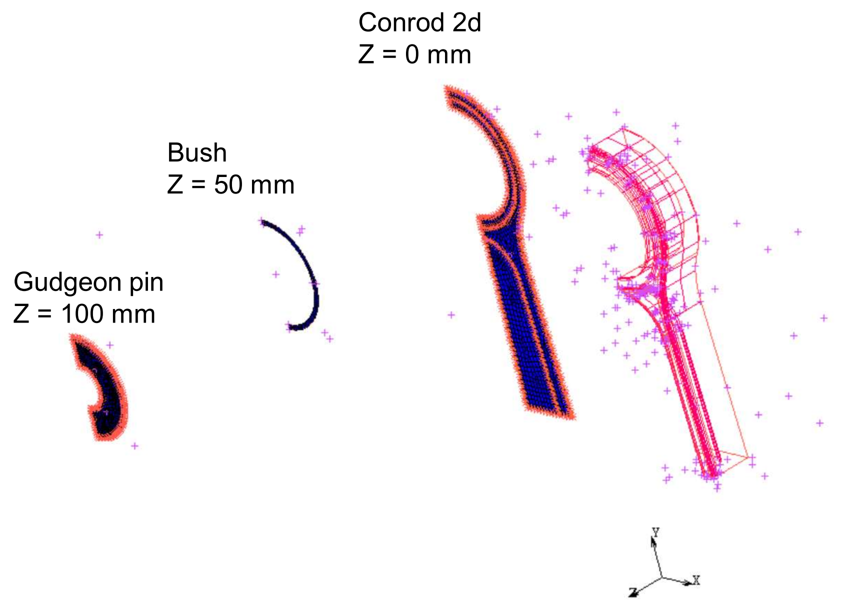
We assume that the components are in steel. The material is assumed to be homogeneous and isotropic.

$E = 210000$ MPa
$\nu = 0.3$

Mesh generation: MOVE
Components positioning

```
|-----------------------------------
|   PLOT THE BUSH AND GUDGEON PIN
|   ELEMENTS AND CURVES
|-----------------------------------
*invisible_all_sets
*visible_set bush
*visible_set bush_elems
*visible_set gudgeon_pin
*visible_set gudgeon_pin_elems
|-----------------------------------
|   MESH GENERATION: MOVE (STEP 1)
|   BOTH BUSH AND GUDGEON PIN
|-----------------------------------
*move_reset
*set_move_translation x 50
*set_move_translation z 50
*move_combined
all_visible
*invisible_set bush
*invisible_set bush_elems
|-----------------------------------
|   MESH GENERATION: MOVE (STEP 2)
|   GUDGEON PIN ONLY
|-----------------------------------
*move_combined
all_visible
*visible_all_sets
```

The components will be aligned at X=0 at different Z, using the function MOVE COMBINED.



Mesh generation: SWEEP

Components positioning

```
|-----------------------------------
|   MESH GENERATION: SWEEP
|-----------------------------------
*set_sweep_tolerance 0.0001
*sweep_all
|-----------------------------------
|   Command prompt
|   Deleting 8 duplicate nodes!
|   Deleting 0 collapsed elements!
|   Deleting 0 duplicate elements!
|   Deleting 228 duplicate points!
|-----------------------------------
```

We decide to remove the redundant nodes by the function SWEEP at MESH GENERATION MENU without collapsing any elements. The tolerance has been set lower than minimum «element size» (ca. 0.2 mm) divided by 2 (due to the use of quadratic elements), therefore it is set equal to 0.0001 mm.

Save the file at this stage!!!!

Agenda

Goal

The geometry import

Mesh generation

Contact

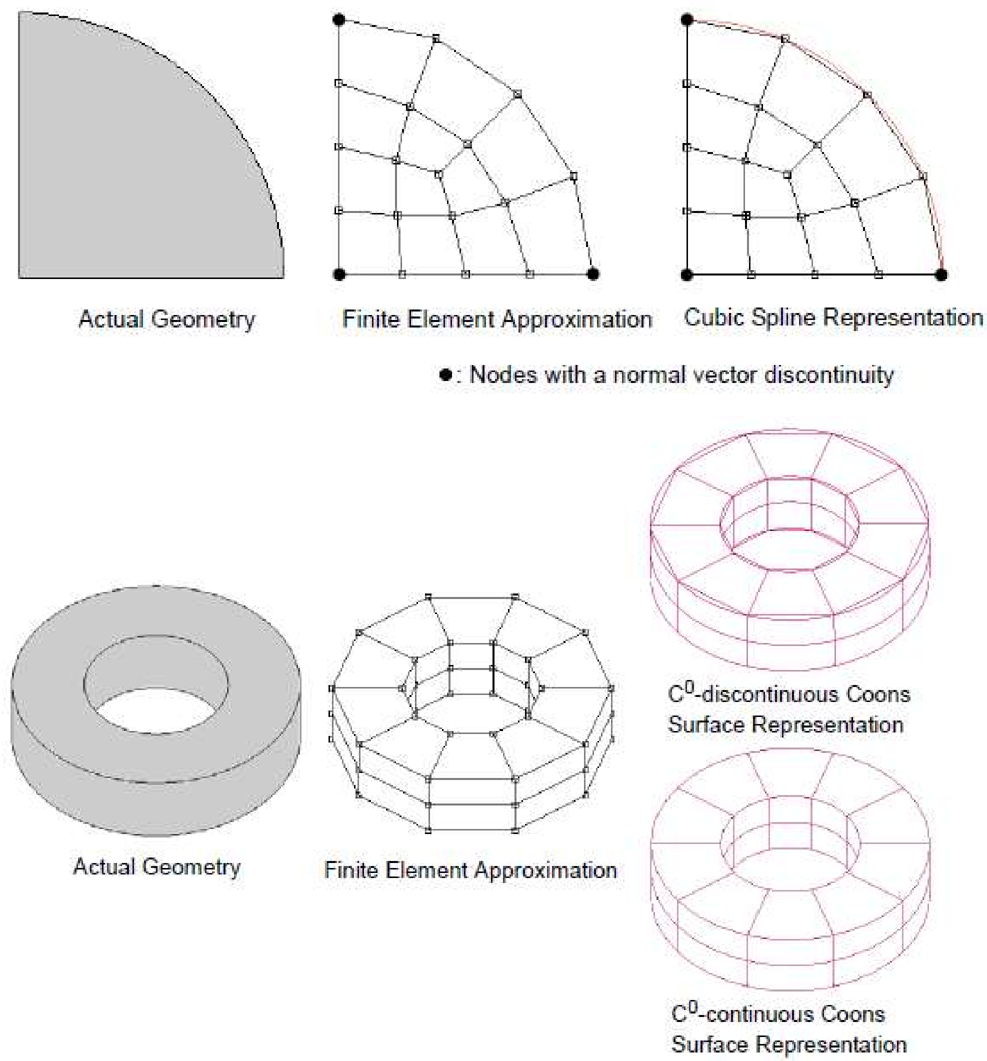
References

Contact

Contact bodies: ANALYTICAL vs DiSCRETE

Planar Model

3D Model



Contact

Contact bodies: ANALYTICAL & DISCONTINUITY

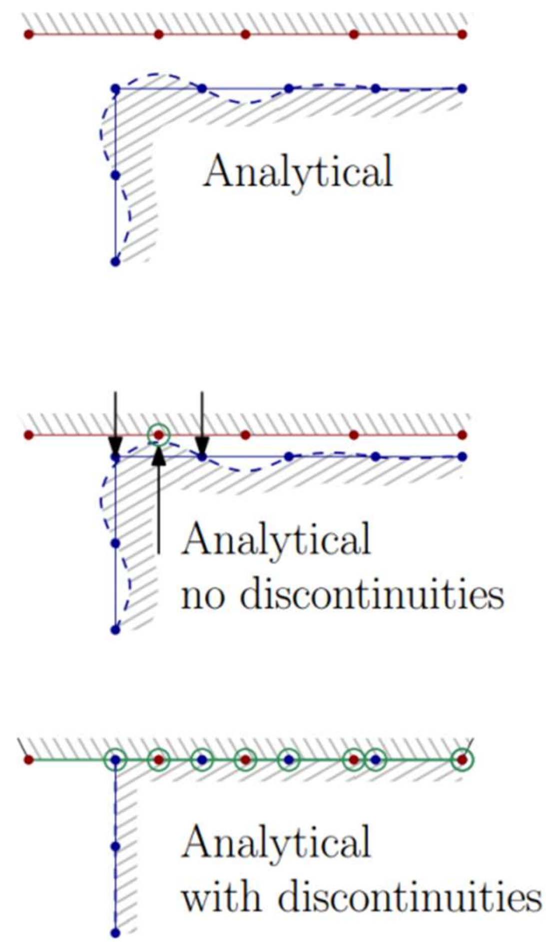
These commands are used to set the type of boundary description for a deformable body or a rigid body with heat transfer, modeled using lower-order finite elements.

In the default discrete description, the boundary of the contacted body is described by the finite elements defining the body. This can cause inaccuracies due to the fact that the normals of the body are not continuous for a curved boundary described with lower-order elements.

In the analytical description, linear segments are replaced by:
- spline curves for 2D contacted bodies;
- Coons surfaces for 3D contacted bodies.

These analytical entities provide a smooth description of the boundary of contacted bodies and nodes of a contacting body are now touching the analytical entities instead of the actual finite elements. The analytical entities are updated as the body is deformed.

Since the modeled structure may have corners (2D) or edges (3D) where the normals are discontinuous, at such places the smoothing procedure should not be applied. They can be identified:
- manually;
- automatically;
- combined manually and automatically.

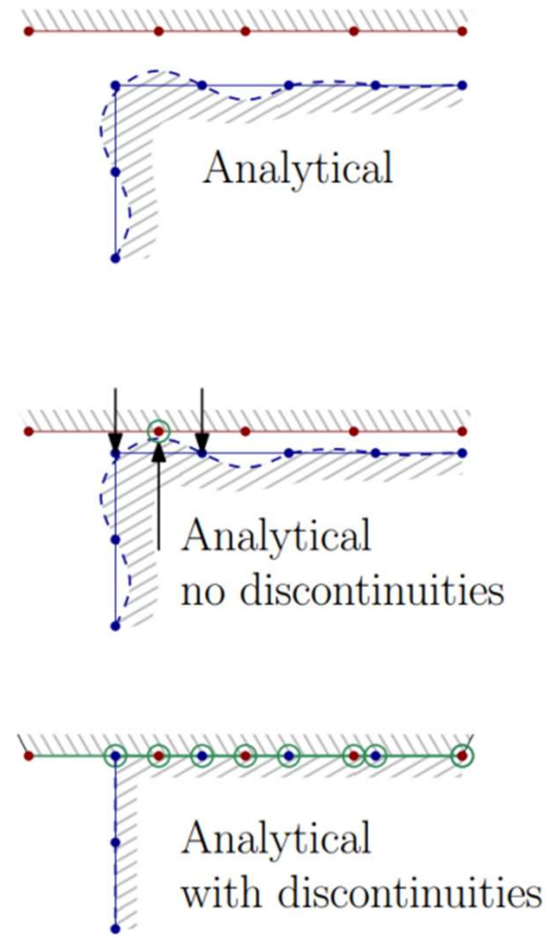


Contact

Contact bodies

When the discontinuity method is manual, one can use the commands `add_cbody_dc_nodes` (2D) and `add_cbody_dc_edges` (3D) to add a list of nodes (2D) or edges (3D) with normal discontinuities. When the discontinuity method is automatic, one can define a threshold angle (default 60 degrees). If the angle between the normals of two adjacent segments exceeds this threshold, MSC.Marc will automatically add the corresponding node or edge to the list of discontinuities. When the discontinuity method is manual and automatic, the user can still manually define a list of discontinuity nodes or edges and MSC.Marc will compare the angle between the normals of adjacent segments with the threshold angle and add nodes or edges to the user-defined list if needed.

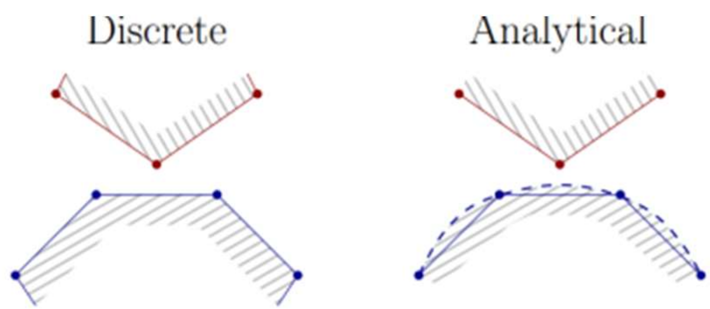
Finally, in 3D one can activate C0-continuity at edges where the normals are discontinuous. If this is done, the in-plane description of the segment is modified, taking into account the curved shape of the edge where the boundary normal is discontinuous.



Contact

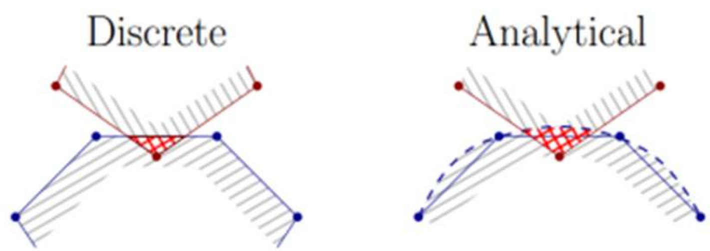
Contact bodies

Time : t_1
Load: $F = F_1$



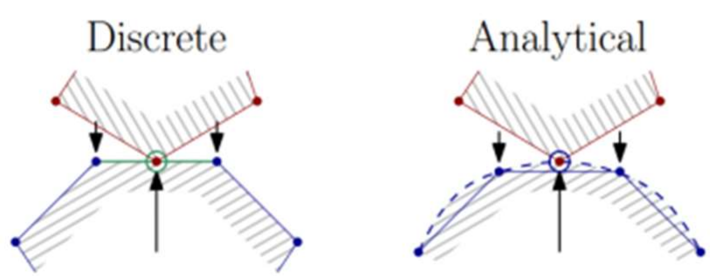
Contact CHECK:
NO contact

Time : t_2 (STEP 0)
Load: $F = F_2 > F_1$



Contact CHECK:
YES contact
The contact is influenced by the body definition
$d_{\text{discrete-discrete}} < d_{\text{discrete-analytical}}$

Time : t2 (STEP 1)
Load: $F = F_2 > F_1$



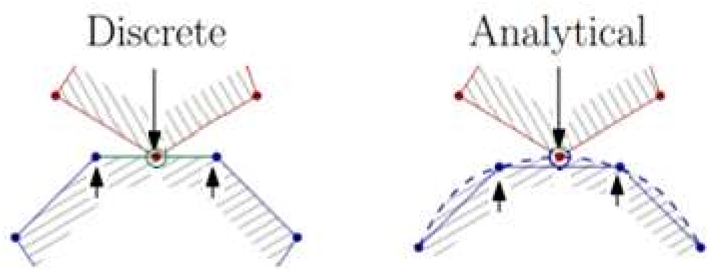
The force of the interaction between the bodies is a repulsive forces, that are balanced at the bodies nodes.

A kinematic relation between forces the nodes of the upper body to lie on the surface of the bottom body since …

Contact

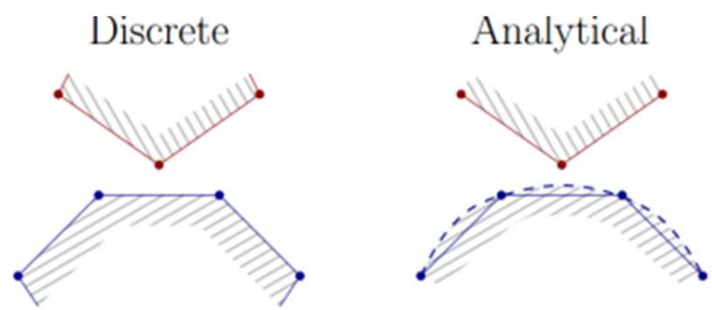
Contact bodies

Time : t3 (STEP 0)
Load: $F = F_1 < F_2$



CHECK CONTACT:
The reaction force of the interaction between the bodies becomes a tensile forces, the node of the upper body is detached from the surface of the lower one.

Time : t3 (STEP 1)
Load: $F = F_1 < F_2$



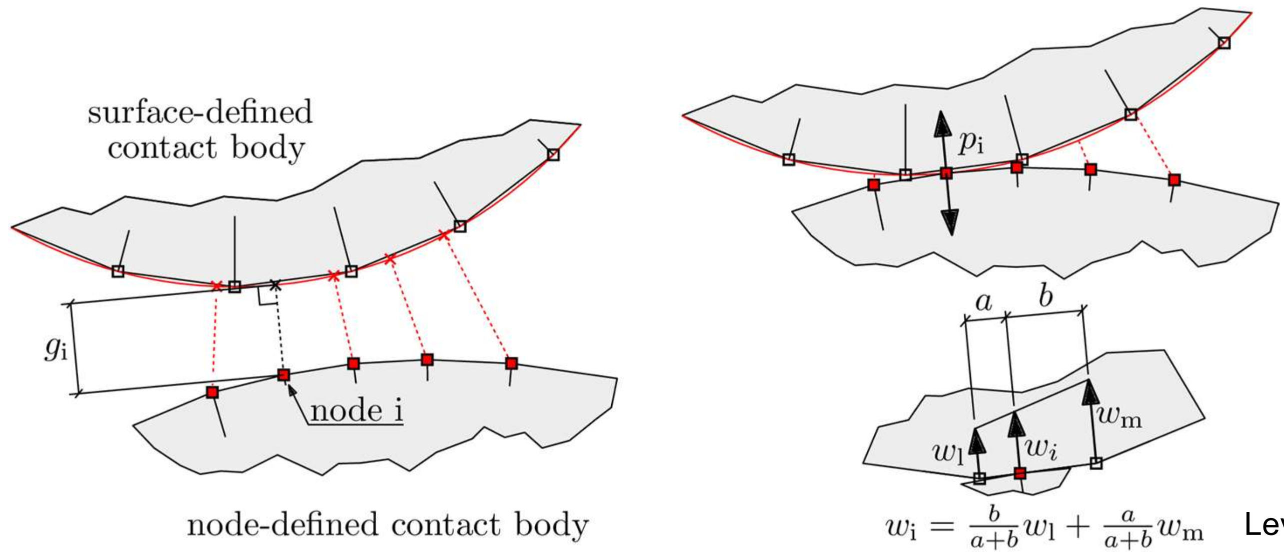
The Signorini inaqualies relations that manage the unilateral contact definition cannot be solved in a FEM equation system; therefore, unilateral BCs are treated by activating or deactivating bilateral BCs.

$$\begin{cases} g_i \geq 0 \\ p_i \geq 0 \\ g_i \cdot p_i \geq 0 \end{cases}$$

Contact

Contact bodies

w_i, w_l, w_m : nodal displacement of the *i*-th, *l*- or *m*-node.
g_i: distance from the *i*-th node to the surface of the other body
p_i: contact pressure



Lever rule

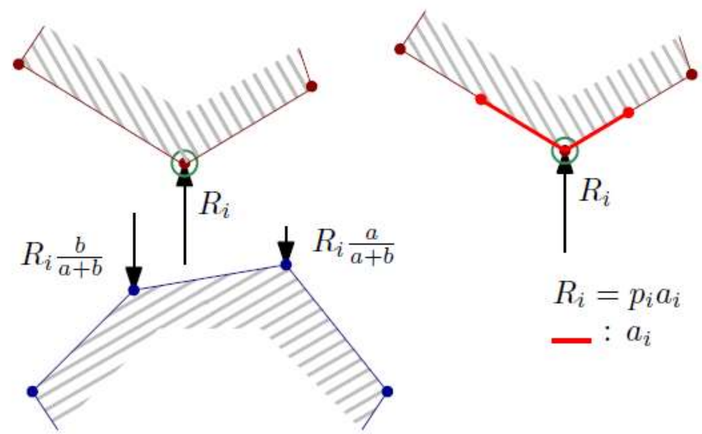
Engineering practice	The NODES are considered from the body with …	The EDGES/SURFACES are considered from the body with ..
1- Bodies with different mesh size	… the finest mesh	… the coarse mesh
2- Bodies with smoothed or notched profiles.	… the notched geometry	… the smoothed geometry

Contact
Contact bodies

The Signorini inaqualies relations that manage the unilateral contact definition cannot be solved in a FEM equation system; therefore, unilateral BCs are treated by activating or deactivating bilateral BCs.

$$\begin{cases} g_i \geq 0 \\ p_i \geq 0 \\ g_i \cdot p_i = 0 \end{cases}$$

The contact pressure distribution is calculated from the reaction forces R_i that act in a prescribed surface a_i over which that forces are distributed. An example on 2D model is reported, where the area on which the force is distributed is defined by the sum of the mid-portion body edges involved in the contact problem.



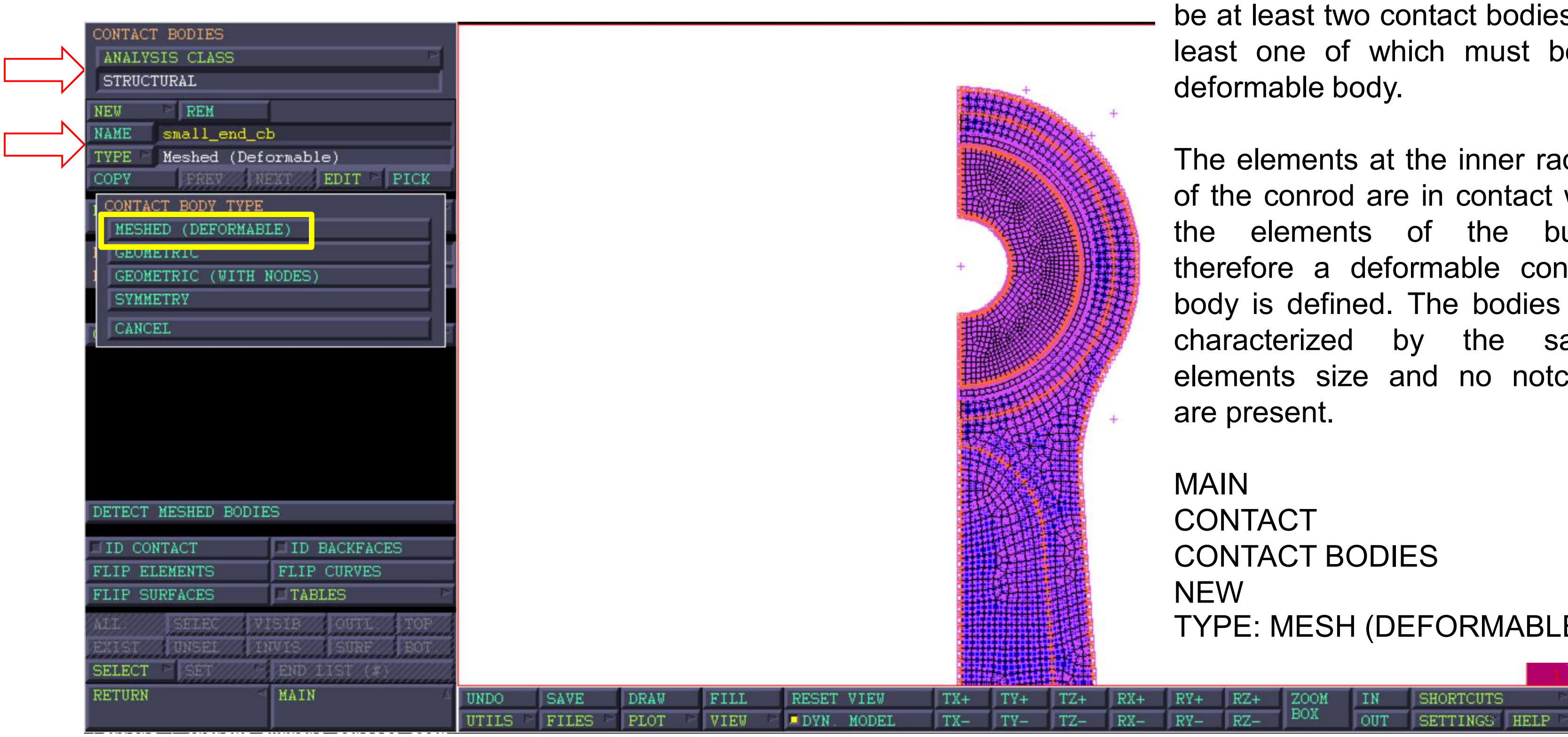
Contact

Contact bodies: summary

Contact body	YES/NO	TYPE	From each body, we consider the …
Shank_elems	NO	-	-
Pocket_elems	NO	-	-
Small_end_elems	YES	DISCRETE	NODES
Bush_elems	YES	ANALITYCAL	SPLINE
Gudgeon_pin_elems	YES	DISCRETE	NODES

Contact

Contact bodies: small_end_cb



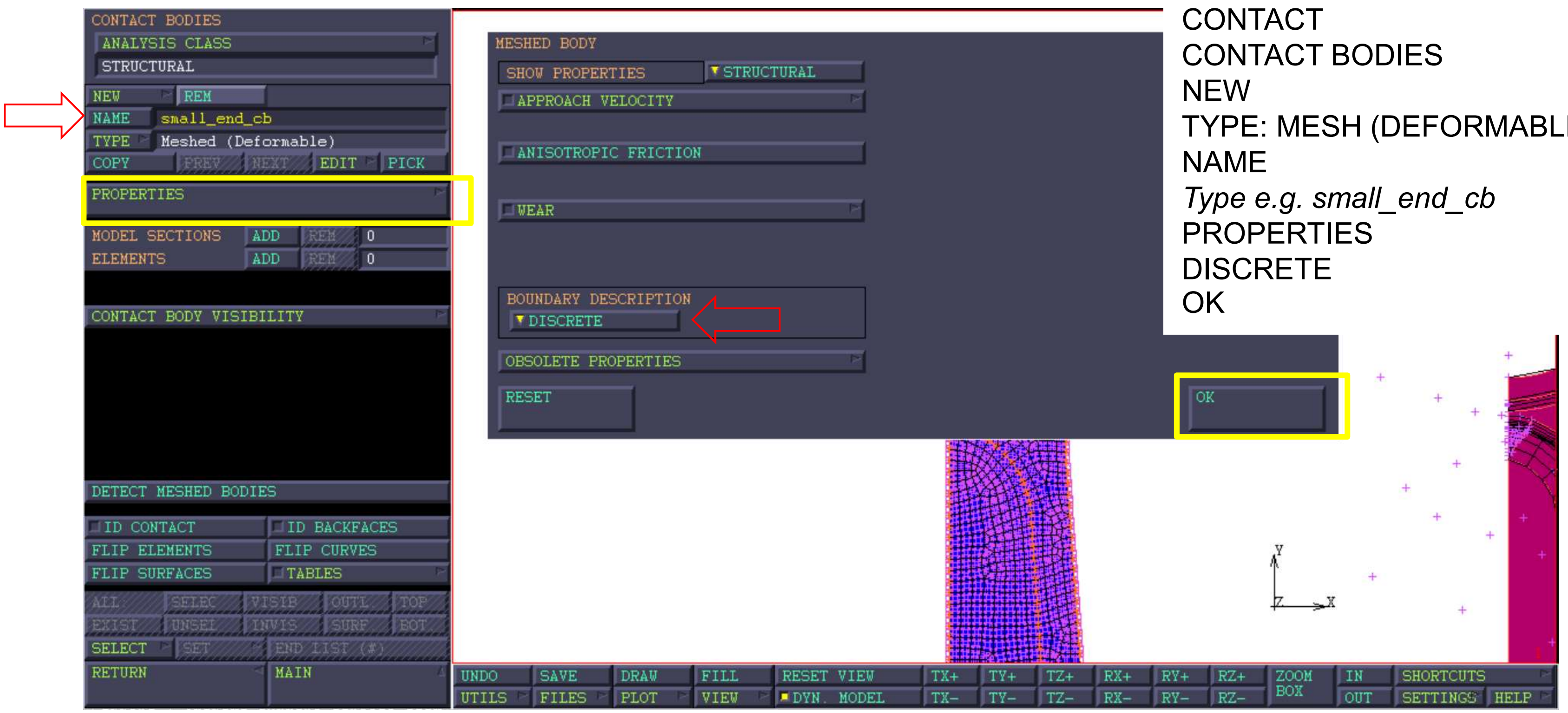
A contact body is a set of curves, surfaces, or **elements** that act as a body in a contact analysis.
In a contact analysis, there must be at least two contact bodies, at least one of which must be a deformable body.

The elements at the inner radius of the conrod are in contact with the elements of the bush, therefore a deformable contact body is defined. The bodies are characterized by the same elements size and no notches are present.

MAIN
CONTACT
CONTACT BODIES
NEW
TYPE: MESH (DEFORMABLE)

Contact

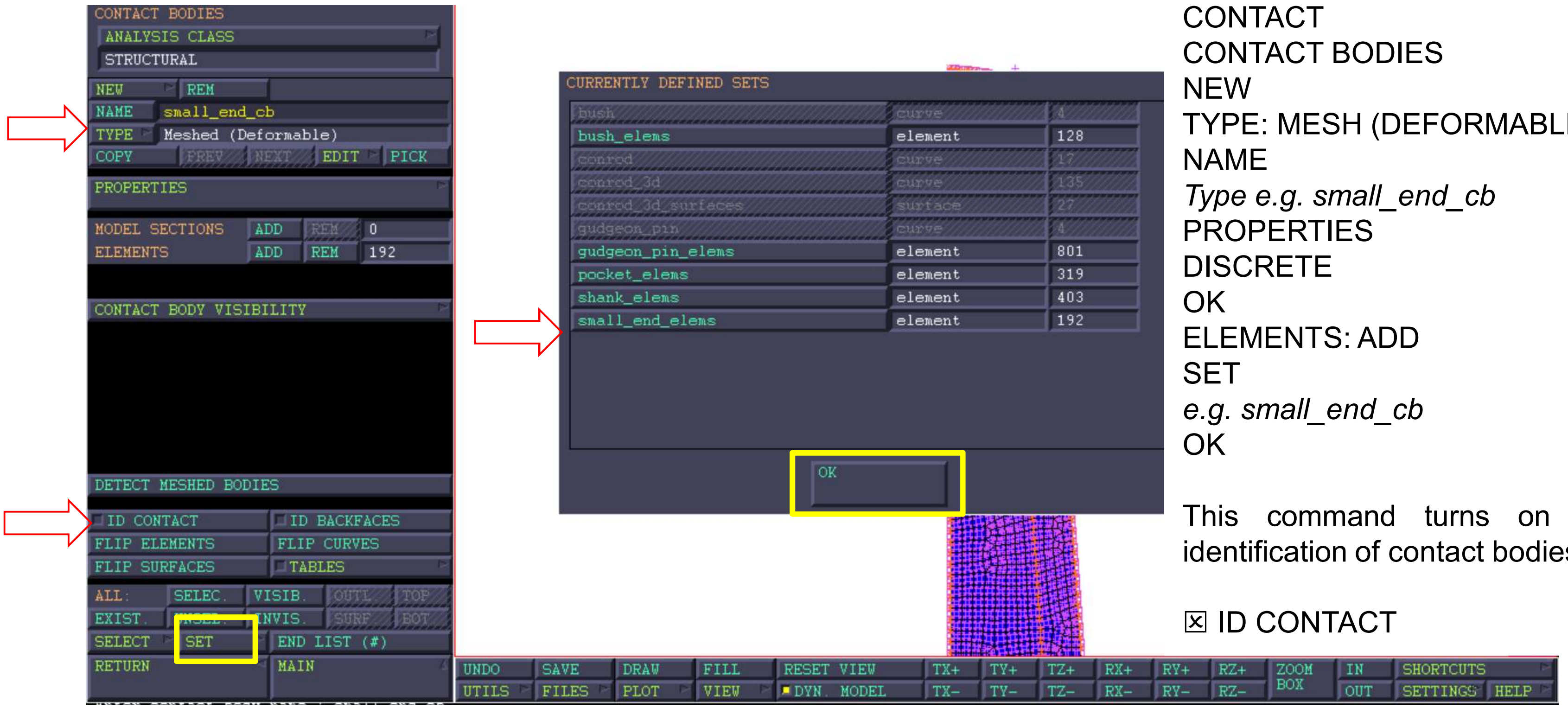
Contact bodies: small_end_cb



MAIN
CONTACT
CONTACT BODIES
NEW
TYPE: MESH (DEFORMABLE)
NAME
Type e.g. small_end_cb
PROPERTIES
DISCRETE
OK

Contact

Contact bodies: small_end_cb



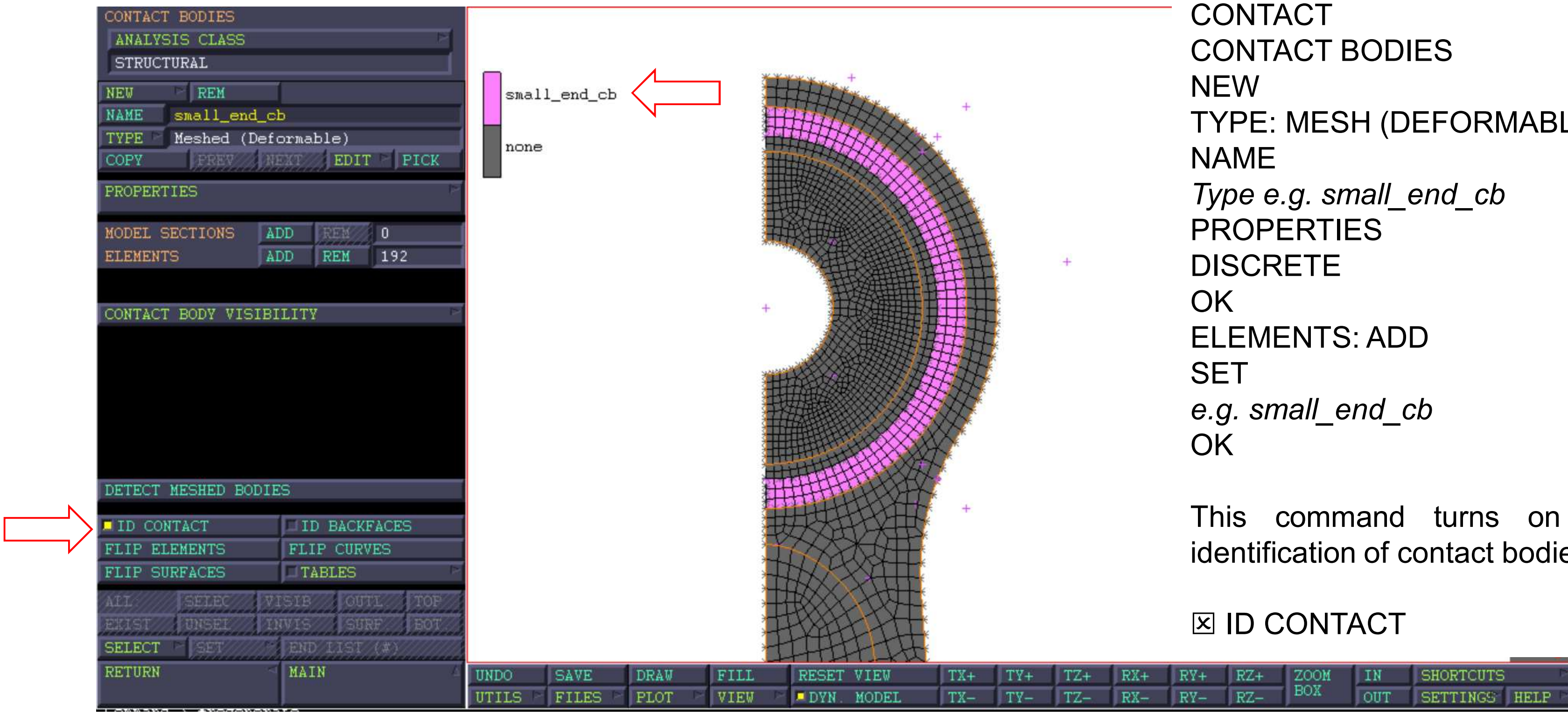
MAIN
CONTACT
CONTACT BODIES
NEW
TYPE: MESH (DEFORMABLE)
NAME
Type e.g. small_end_cb
PROPERTIES
DISCRETE
OK
ELEMENTS: ADD
SET
e.g. small_end_cb
OK

This command turns on the identification of contact bodies.

☒ ID CONTACT

Contact

Contact bodies: small_end_cb



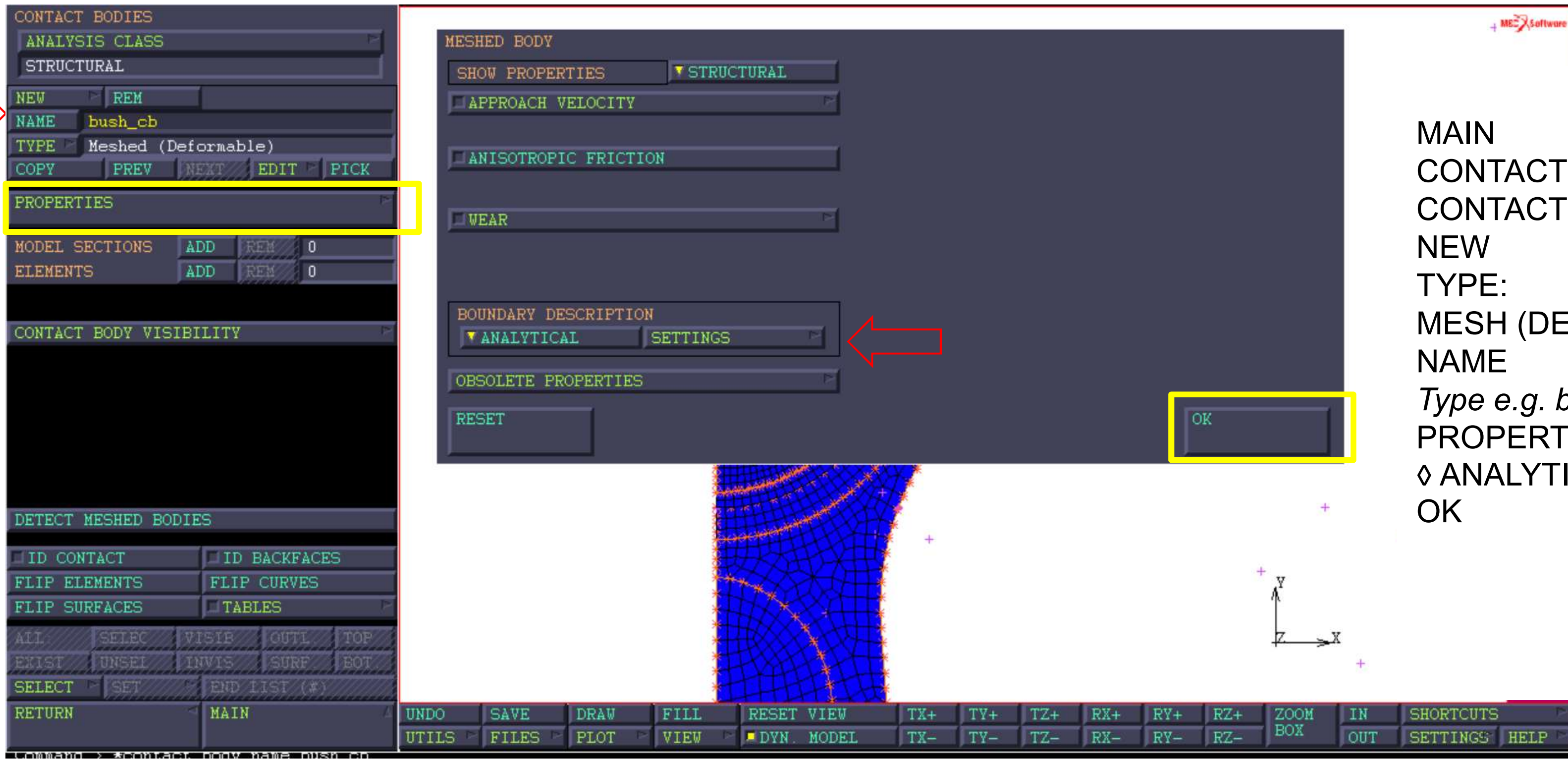
MAIN
CONTACT
CONTACT BODIES
NEW
TYPE: MESH (DEFORMABLE)
NAME
Type e.g. small_end_cb
PROPERTIES
DISCRETE
OK
ELEMENTS: ADD
SET
e.g. small_end_cb
OK

This command turns on the identification of contact bodies.

☒ ID CONTACT

Contact

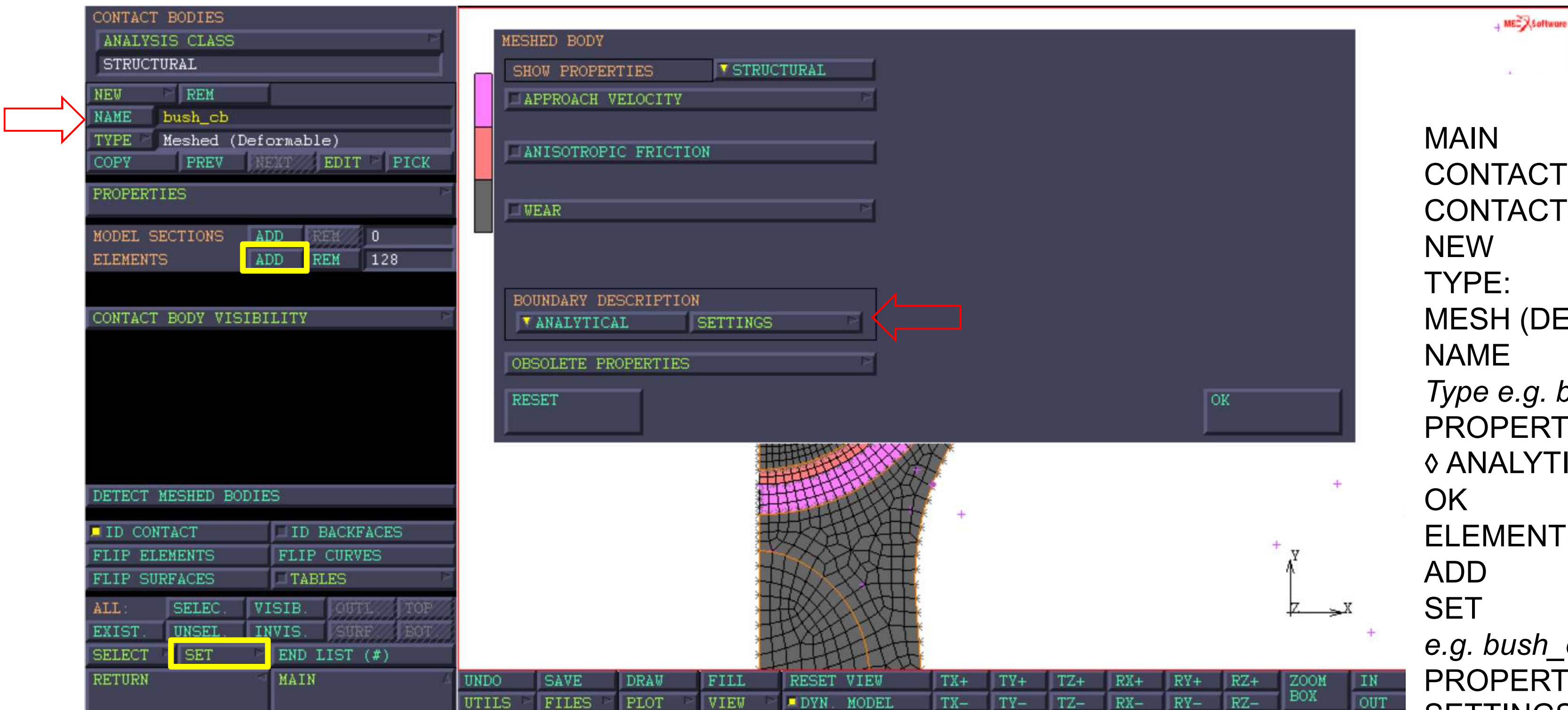
Contact bodies: bush_cb



MAIN
CONTACT
CONTACT BODIES
NEW
TYPE:
MESH (DEFORMABLE)
NAME
Type e.g. bush_cb
PROPERTIES
◊ ANALYTICAL
OK

Contact

Contact bodies: bush_cb



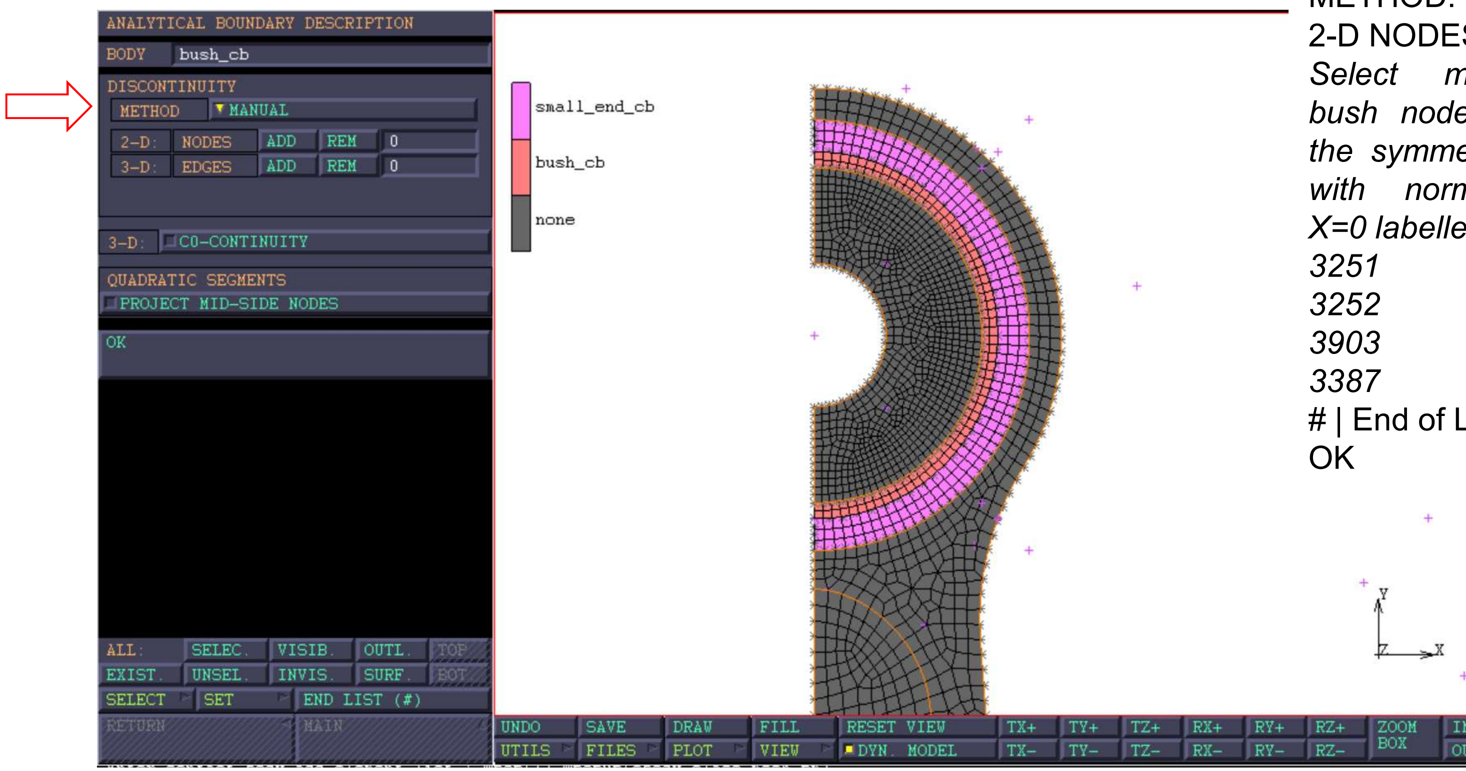
MAIN
CONTACT
CONTACT BODIES
NEW
TYPE:
MESH (DEFORMABLE)
NAME
Type e.g. bush_cb
PROPERTIES
◊ ANALYTICAL
OK
ELEMENTS
ADD
SET
e.g. bush_elems
PROPERTIES
SETTINGS

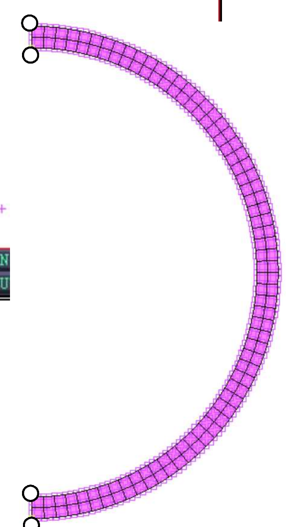
Contact

Contact bodies: bush_cb

To delete at the analytical body bush the discontinuity nodes, see as follows:

DISCONTINUITY
METHOD: ◊ MANUAL
2-D NODES: ADD
Select manually the bush nodes located at the symmetry plane ZY with normal direction X=0 labelled as:
3251
3252
3903
3387
| End of List
OK





Contact

Contact bodies: gudgeon_pin_cb

```
|----------------------------------
|   CONTACT BODY
|   BODY 3--> Gudgeon_pin_cb:   Deformable Discrete
|----------------------------------
*new_cbody mesh
*contact_option state:solid
*contact_option skip_structural:off
*contact_body_name
gudgeon_pin_cb
*contact_option defo_desc:discrete
*add_contact_body_elements
gudgeon_pin_elems
|----------------------------------
*identify_contact *regen
*identify_none *regen
|----------------------------------
```

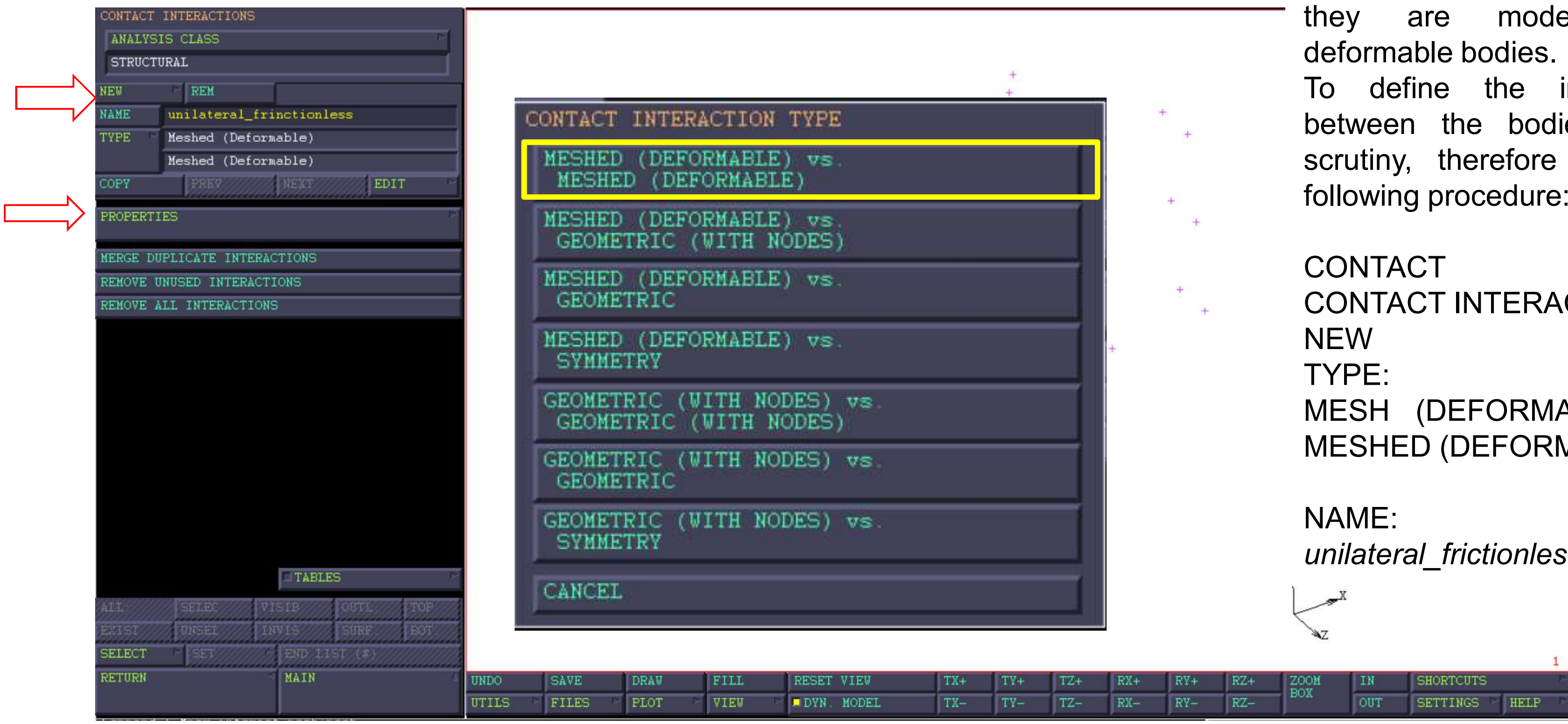
The gudgeon pin is treated as a DISCRETE contact body, as discussed similarly to the `small_end_cb.`
Finally, the contact bodies defined in the model are highlighted by turning on:

☒ ID CONTACT

Contact

Contact interactions

This menu allows for the input of properties defining the interaction between contact bodies (discretized by mesh or defined by geometric entities such as curves or surfaces). Typical examples of such properties are the coefficient of friction, the heat transfer coefficient and the contact separation stress. Contact interaction sets are referenced by using the contact table option.



Our parts are meshed and they are modelled as deformable bodies.

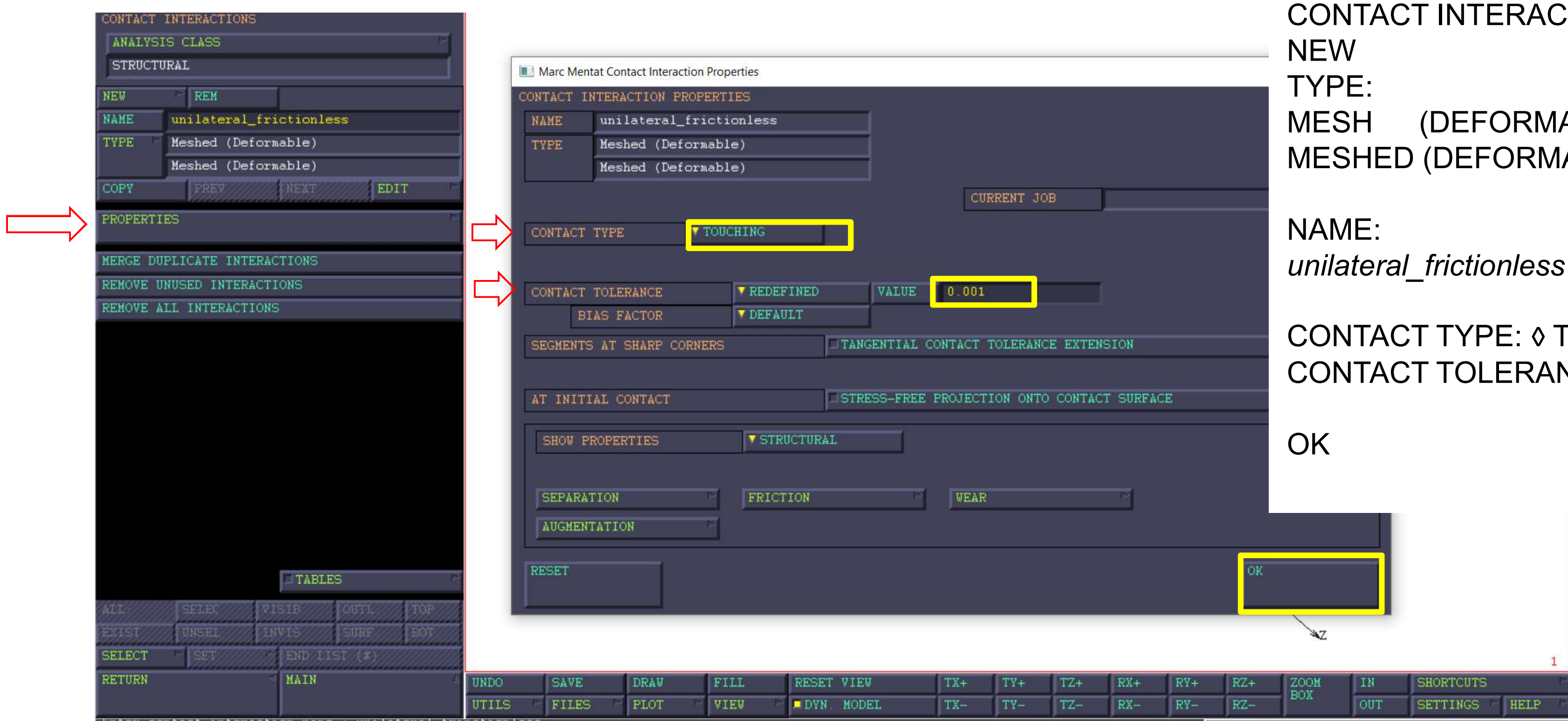
To define the interaction between the bodies under scrutiny, therefore use the following procedure:

CONTACT
CONTACT INTERACTIONS
NEW
TYPE:
MESH (DEFORMABLE) vs MESHED (DEFORMABLE)

NAME: Type *unilateral_frictionless*

Contact

Contact interactions



To define the interaction between the bodies under scrutiny:

CONTACT
CONTACT INTERACTIONS
NEW
TYPE:
MESH (DEFORMABLE) vs MESHED (DEFORMABLE)

NAME: Type *unilateral_frictionless*

CONTACT TYPE: ◊ TOUCHING
CONTACT TOLERANCE: 0.001

OK

Contact
Contact table

A contact table is a set of entries which specifies the relationship between contact bodies in a contact analysis.

Contact tables can be used for different purposes, like:

- indicate which set of bodies may or may not touch each other, so that computational time can be saved;
- define different properties per set of contact bodies, like friction coefficient, error tolerance, separation force, and film coefficient;
- activate glued contact, which can be effectively used to couple separately meshed parts of a structure.

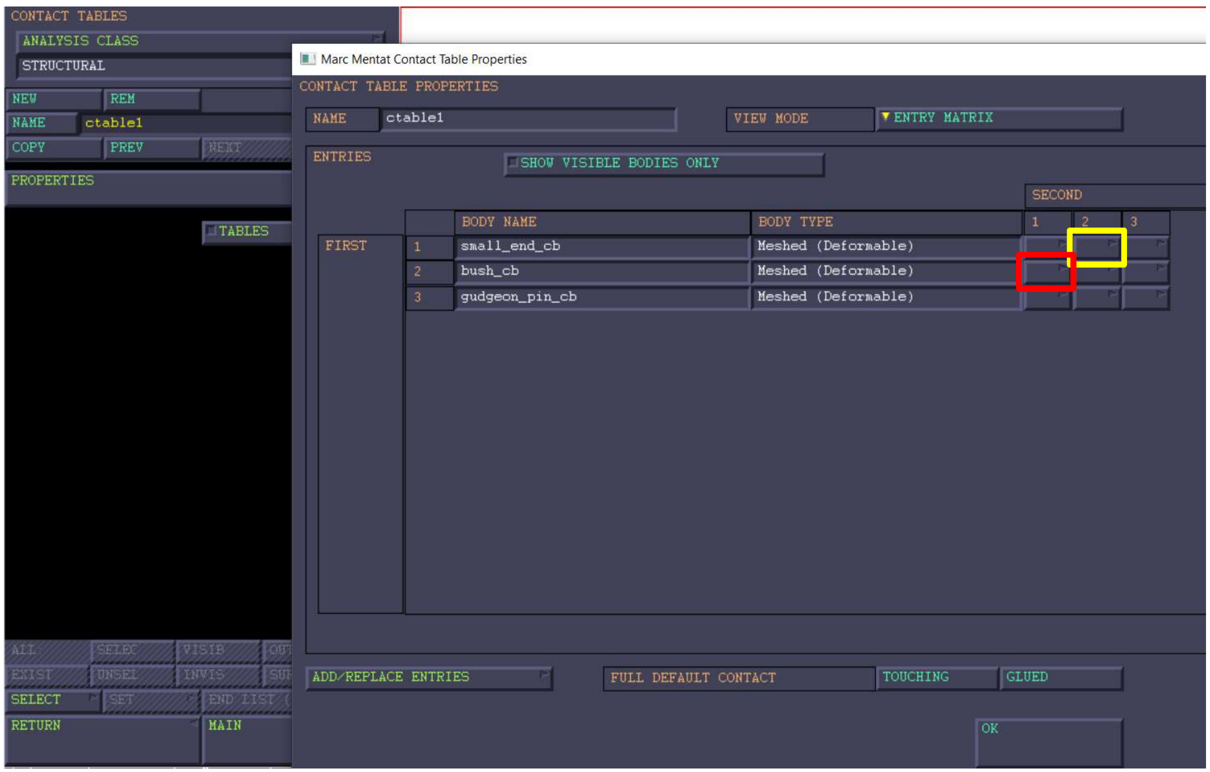
Note that the contact tables must be activated in the loadcase where they are to be used. This is defined in the CONTACT menu for the different loadcase types. Notice that if the user wants to deactivate existing contact between bodies, only selecting a different contact table in which contact between the relevant bodies is not allowed is not sufficient, you also have to release contact between those bodies.

For the correct detection of initial contact (before the first loadcase), the contact table should also be activated in the current job. This is done in the INITIAL CONTACT menu in the CONTACT CONTROL menu for each analysis class (e.g. for an uncoupled structural analysis, it is defined in JOBS-> PROPERTIES-> CONTACT CONTROL-> INITIAL CONTACT-> CONTACT TABLE).

By default, if no contact table is used, every deformable body detects possible contact with every other body including itself.

Contact

Contact table



To define the contact between the bodies under scrutiny:

CONTACT
CONTACT TABLES
NEW
NAME: *e.g.* Type *ctable_interference _fit*
PROPERTIES
ACTIVE

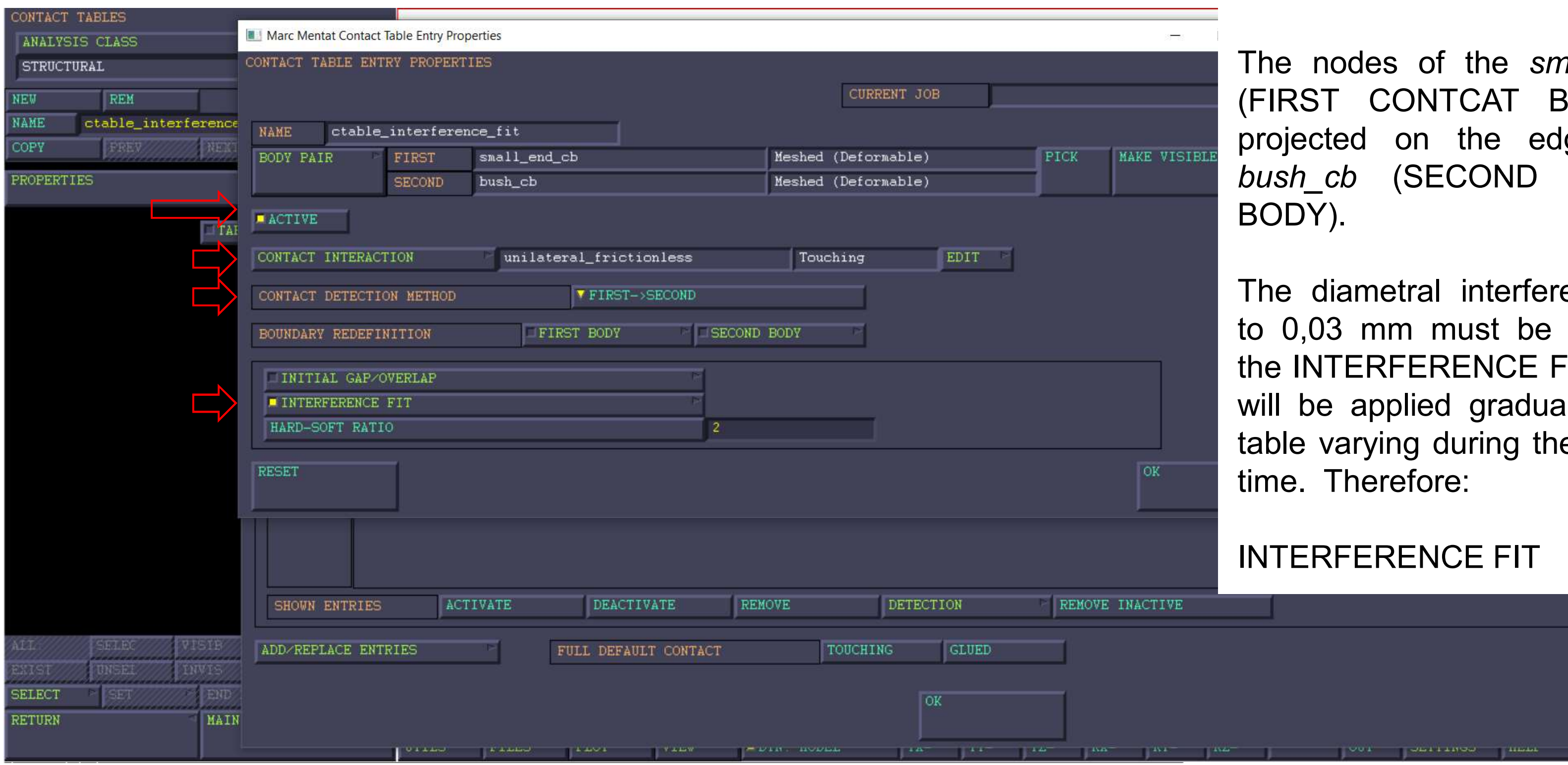
The table is defined as a battleship; where the contact bodies are listed. (FIRST BODY : column; SECOND BODY: row)

To define the contact between the *small_end_cb* and the *bush_cb* select or the box highlighted in yellow or the box highlighted in red, as shown in Figure.

For the present model, the yellow box has been considered, and the red has been inherited.

Contact

Contact table



CONTACT INTERACTION
Upload the *unilateral_frictionless* contact interaction defined previously

CONTACT DETECTION METHOD:
FIRST→ SECOND

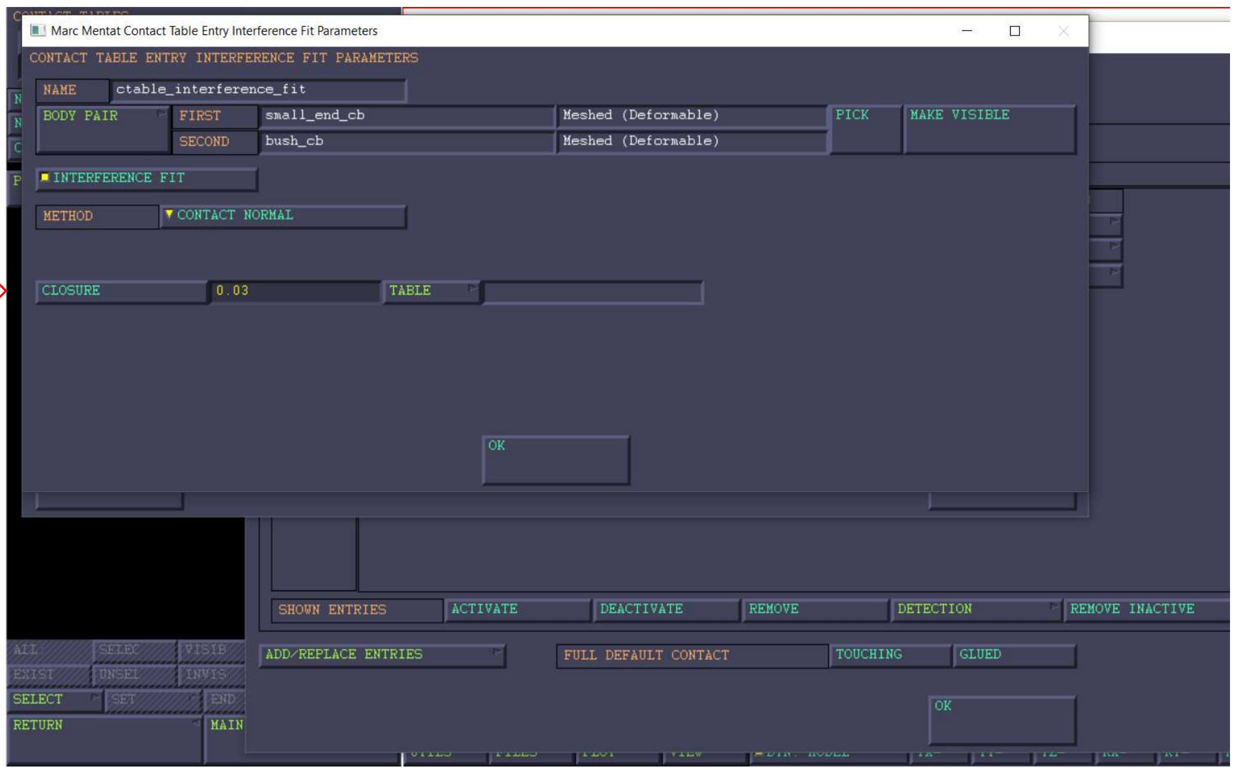
The nodes of the *small_end_cb* (FIRST CONTCAT BODY) are projected on the edge of the *bush_cb* (SECOND CONTACT BODY).

The diametral interference equal to 0,03 mm must be defined by the INTERFERENCE FIT menu. It will be applied gradually using a table varying during the modelling time. Therefore:

INTERFERENCE FIT

Contact

Contact table



CONTACT INTERACTION
Upload the *unilateral_frictionless* contact interaction defined previously

CONTACT DETECTION METHOD:
FIRST→ SECOND

The nodes of the *small_end_cb* (FIRST CONTACT BODY) are projected on the edge of the *bush_cb* (SECOND CONTACT BODY).

The diametral interference equal to 0,03 mm must be defined by the INTERFERENCE FIT menu. It will be applied gradually using a table varying during the modelling time. Therefore:

INTERFERENCE FIT

CLOSURE: 0.03

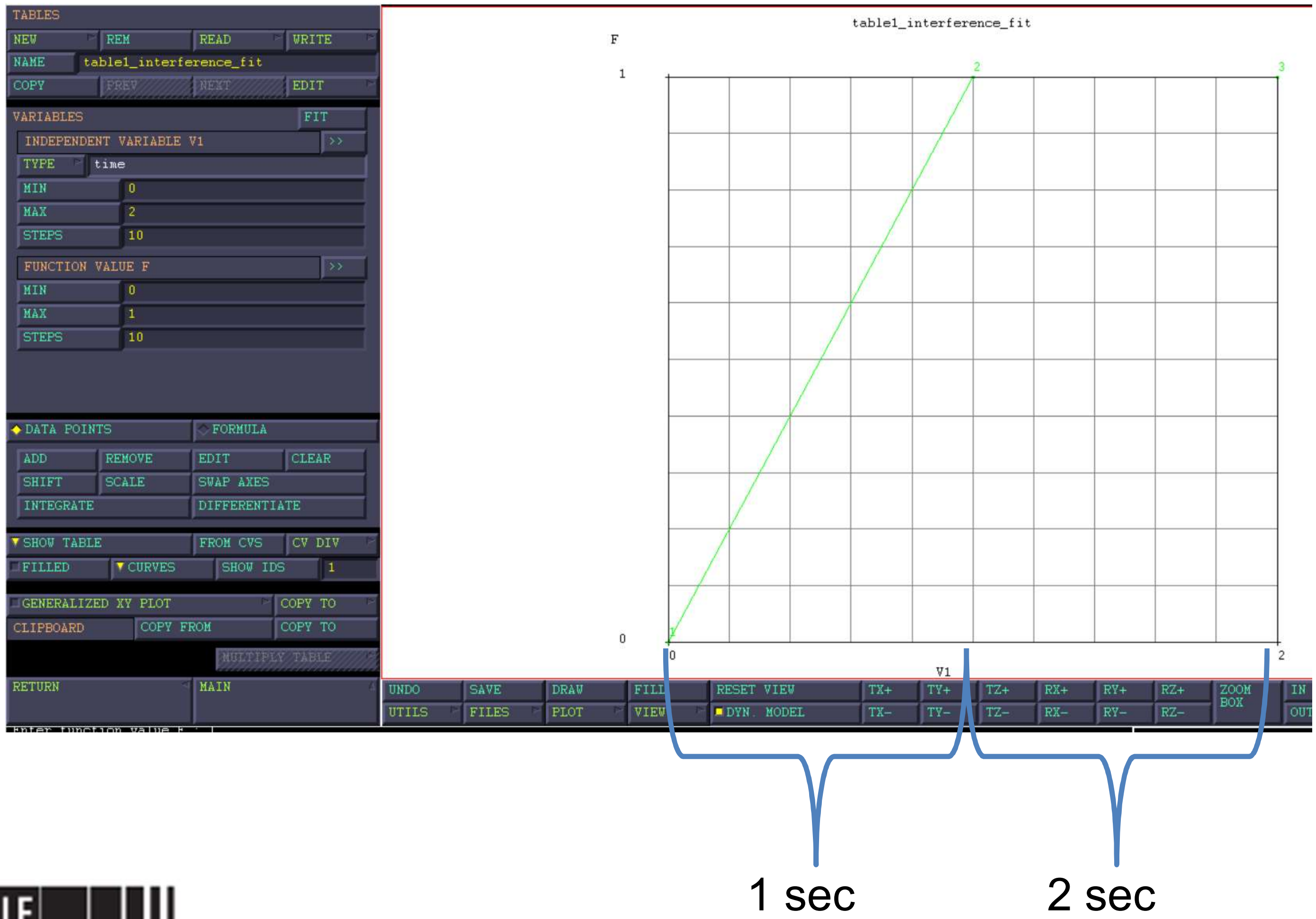
To define the table, we move to the TABLE MENU located at the CONTACT TABLE main menu. Therefore, quit these sub-menus by confirming OK three times.

Contact

Contact table

Contact

Contact table



The TABLE that modulates the INTERFERENCE is defined by DATA POINTS, and the modelling time is set equal to 2, as follows

NAME:
table1_interference_fit
INDIPENDENT VARIABLE V1:
Time
MIN: 0
MAX: 2

DATA POINTS:
0 0
1 1
2 1

The interference is associated to the variable V1 and its final amount (0.03 mm) is applied by this piece-wise law varying during the modelling TIME.
In the first modelling phase (1 sec) the interference is applied with a linear law, during the second phase (2 sec) the interference will be maintained constant at its maximum value.

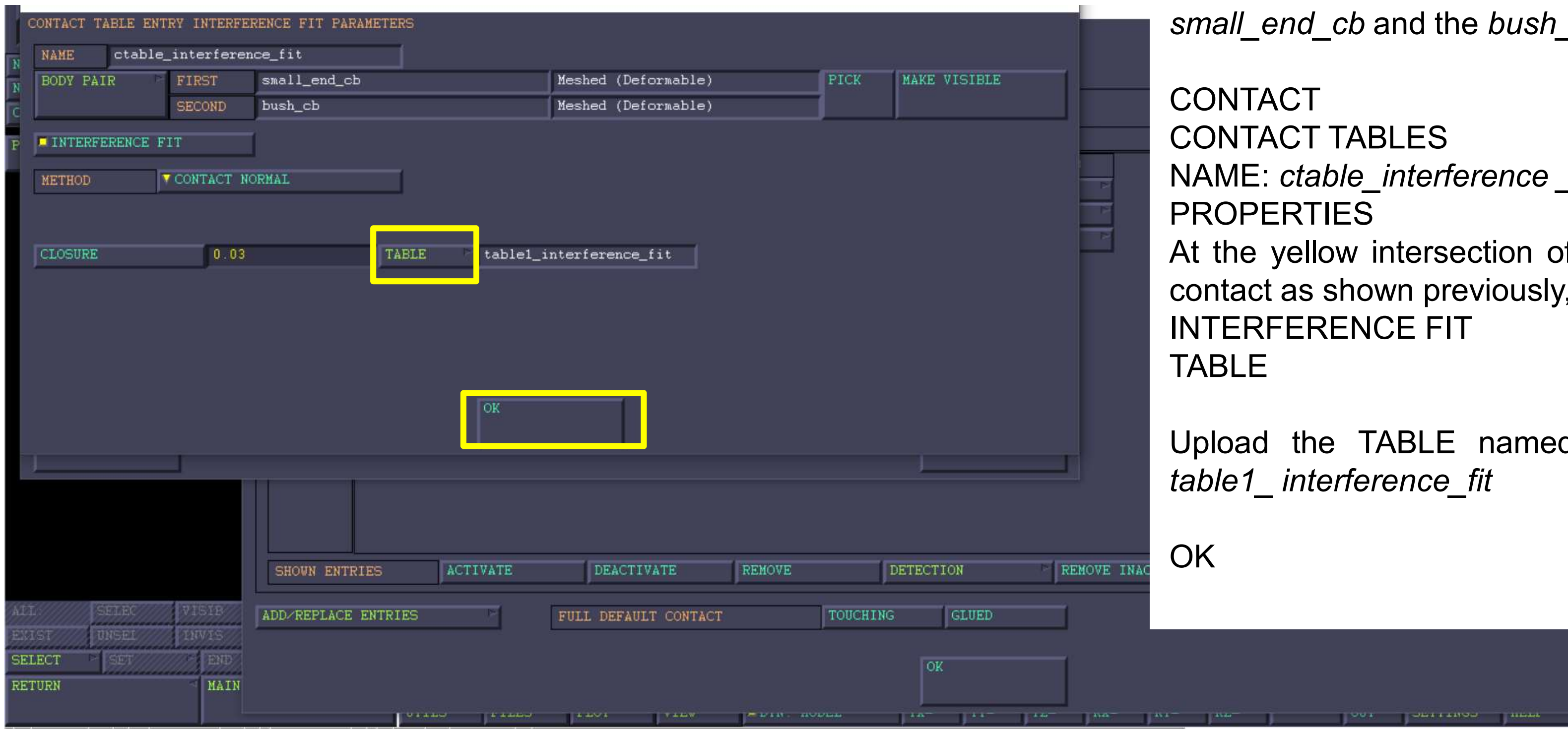
Contact

Contact table

The number of independent and dependent variables must be given.
A table may have multiple independent variables.
The number of independent variables ranges from 1 to 4, each variable having a different table type (physical meaning).
A table consists of a row of data points for each independent variable, and a matrix of function values defining the dependent variable.
Especially for use in the EXPERIMENTAL DATA FIT menus, tables with 1 independent variable and 2 dependent variables may be created. In that case, the table consists of one row of data points for the independent variable, and 2 matrices of function values defining the dependent variables.
Possible combinations of independent and dependent variables:
nindep = 1, ndep = 1: z = f(v1)
nindep = 2, ndep = 1: z = f(v1, v2)
nindep = 3, ndep = 1: z = f(v1, v2, v3)
nindep = 4, ndep = 1: z = f(v1, v2, v3, v4)
nindep = 1, ndep = 2: z = f(v1), z2 = f2(v1)
A table may be applied to parameters specified by the user. Possible parameters include degree of freedom values in boundary conditions, and material property values. Multiple tables may be defined and are stored in the list of currently defined tables.

Contact

Contact table



Now, assess the table to the contact defined between the *small_end_cb* and the *bush_cb.*

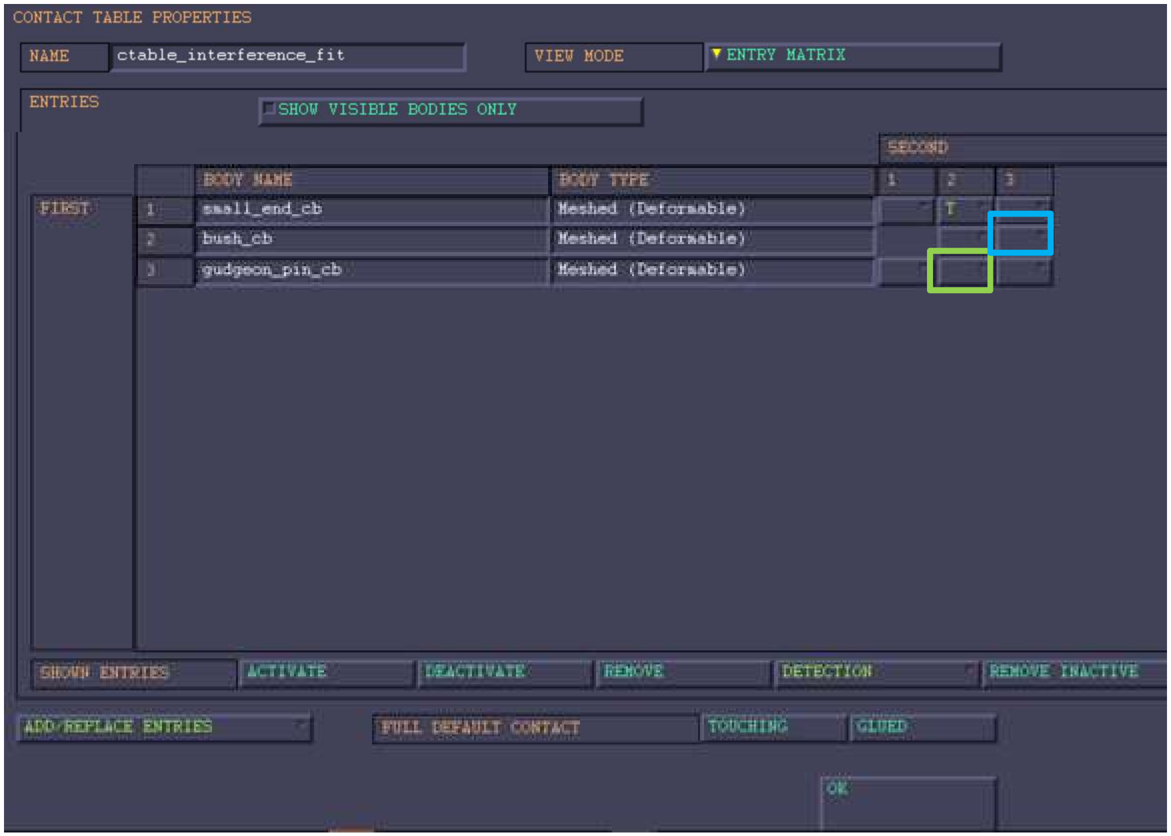
CONTACT
CONTACT TABLES
NAME: *ctable_interference _fit*
PROPERTIES
At the yellow intersection of the contact as shown previously, T
INTERFERENCE FIT
TABLE

Upload the TABLE named as *table1_ interference_fit*

OK

Contact

Contact table



Now, we define the CONTACT between the *bush_cb* and the *gudgen_pin_cb*. The gudgeon pin is not involved during the bush press fit with the conrod small end. However, the gudgeon pin is included into the model and the model is set so that the gudgeon pin does not contribute to the press fit manufacturing phase.

To define the contact between the *bush_cb* and the *gudgeon_pin_cb* select or the box highlighted in green or the box highlighted in blue, as shown in Figure.

For the present model, the green box has been considered, and the blue has been inherited.

The table is defined as a battleship; where the contact bodies are listed.
(FIRST BODY:
column → *gudgeon_pin_cb*;
SECOND BODY:
row → *bush_cb*)

CONTACT
CONTACT TABLES
NAME: *ctable_interference _fit*
PROPERTIES
Select the green box.

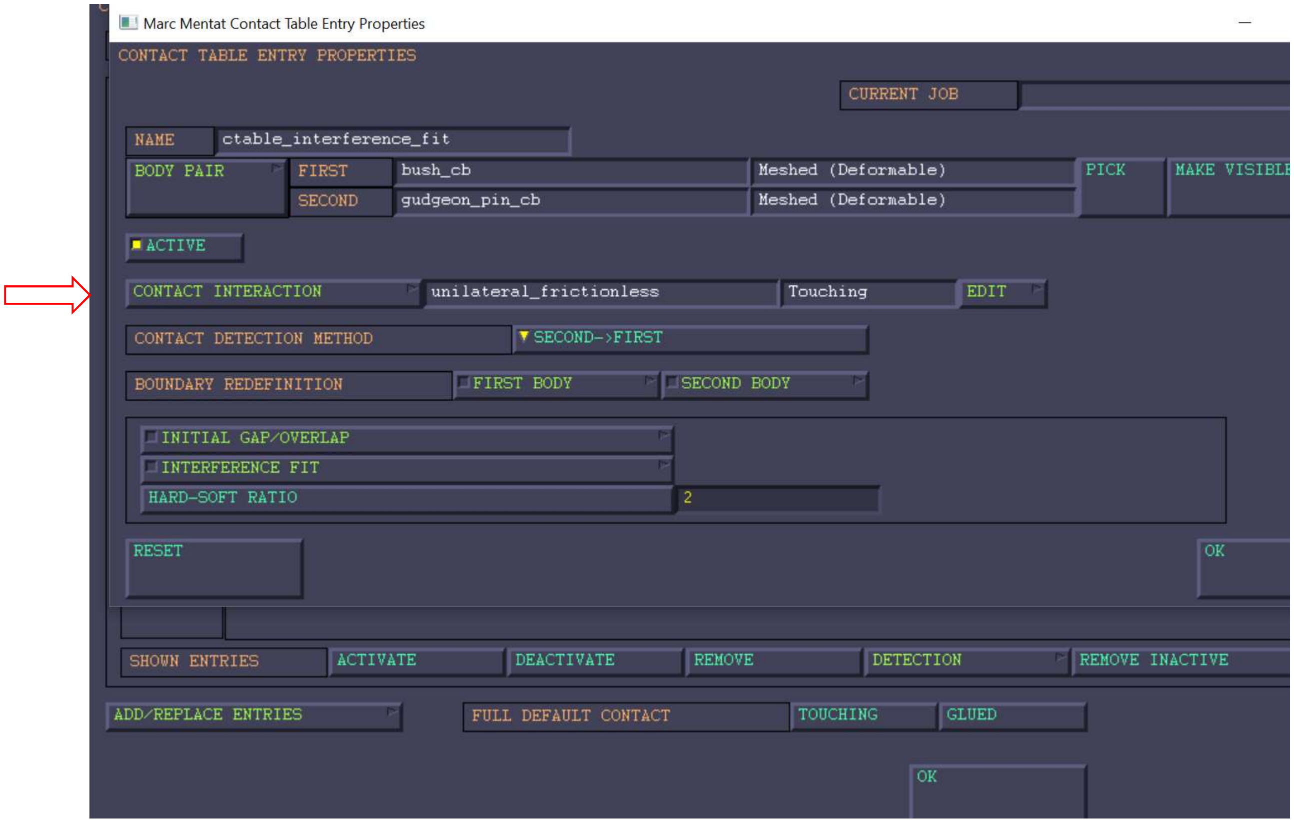
Contact

Contact table

CONTACT
CONTACT TABLES
NAME:
ctable_interference _fit
PROPERTIES
Select the green box.
◊ ACTIVE

Contact

Contact Table



CONTACT INTERACTION
Upload the *unilateral_frictionless* contact interaction previously defined.

CONTACT DETECTION METHOD: ◊ SECOND → FIRST

OK

The nodes of the *gudgeon_pin_cb* (fine mesh) are projected to the edges of the *bush_cb* (coarse mesh).

The contact is frictionless, and the gudgeon pin is mounted as floating, therefore and initial clearence between the bush and the pin will be considered in the following.

As previoulsy discussed the gudgeon pin is not involved during the bush press fit threfore it contact might be deactivate.

Now Deflag ACTIVE!!!!!

Contact

Contact table

CONTACT INTERACTION
Upload the *unilateral_frictionless* contact interaction previously defined.

CONTACT DETECTION METHOD: ◊ SECOND → FIRST

OK

The nodes of the *gudgeon_pin_cb* (fine mesh) are projected to the edges of the *bush_cb* (coarse mesh).

The contact is frictionless, and the gudgeon pin is mounted as floating, thereforeand initial clearence between the bush and the pin itself will be considered in the following.

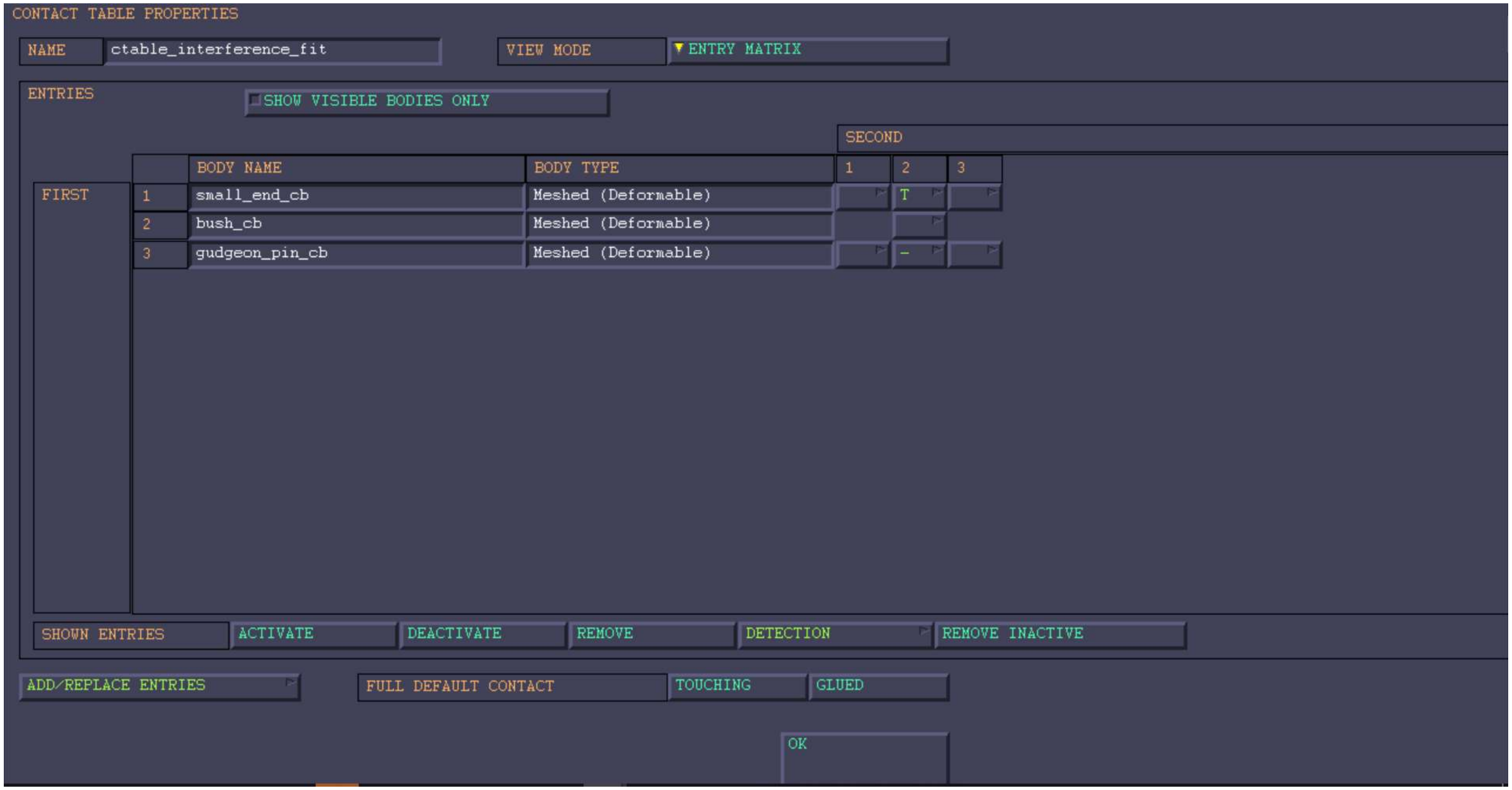
As previoulsy discussed the gudgeon pin is not involved during the bush press fit threfore it contact might be deactivate.

Now Deflag ACTIVE!!!!!

OK

Contact table

Contact Table: Summary



Agenda

Goal

The geometry import

Mesh generation

Contact

References

References

Manufacturing process:

- Shrink-fit

https://www.youtube.com/watch?v=US6rMtLR6nE

- Press-fit

https://www.youtube.com/watch?v=c16bHqs3J2Q

Book:

Strozzi A, Costruzioni di Macchine, Ed. Pitagora (1998):

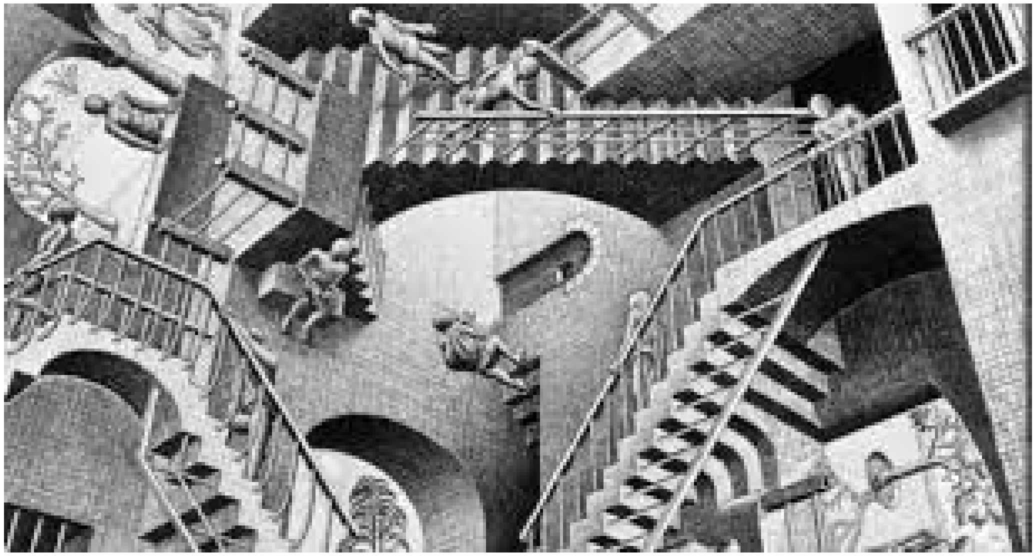
- Plane stress and plane strain, pp. 138-157;
- Pressure vessels, pp. 657-679;
- Shaft-hub press-fit, pp. 690-700;
- Contact problems, pp. 501-518.

Maxima:

- pf_lame.wxmx → analytical evalution of the contact pressure between the bush and the conrod small-end.

FE model file:

- piede_cdm2_geom2d3d_set.mfd → starting file
- conrod2d_rev03.proc → file procedure
- id26_quad8elems_planestress_fullintegration.pdf → element ID 26 formulation
- id3_quad4elems_planestress_fullintegration.pdf → element ID 3 formulation



Escher

… to be continued.