



**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](mailto: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 occurring 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.



# The geometry import

Connecting rod and bush: interference loadcase

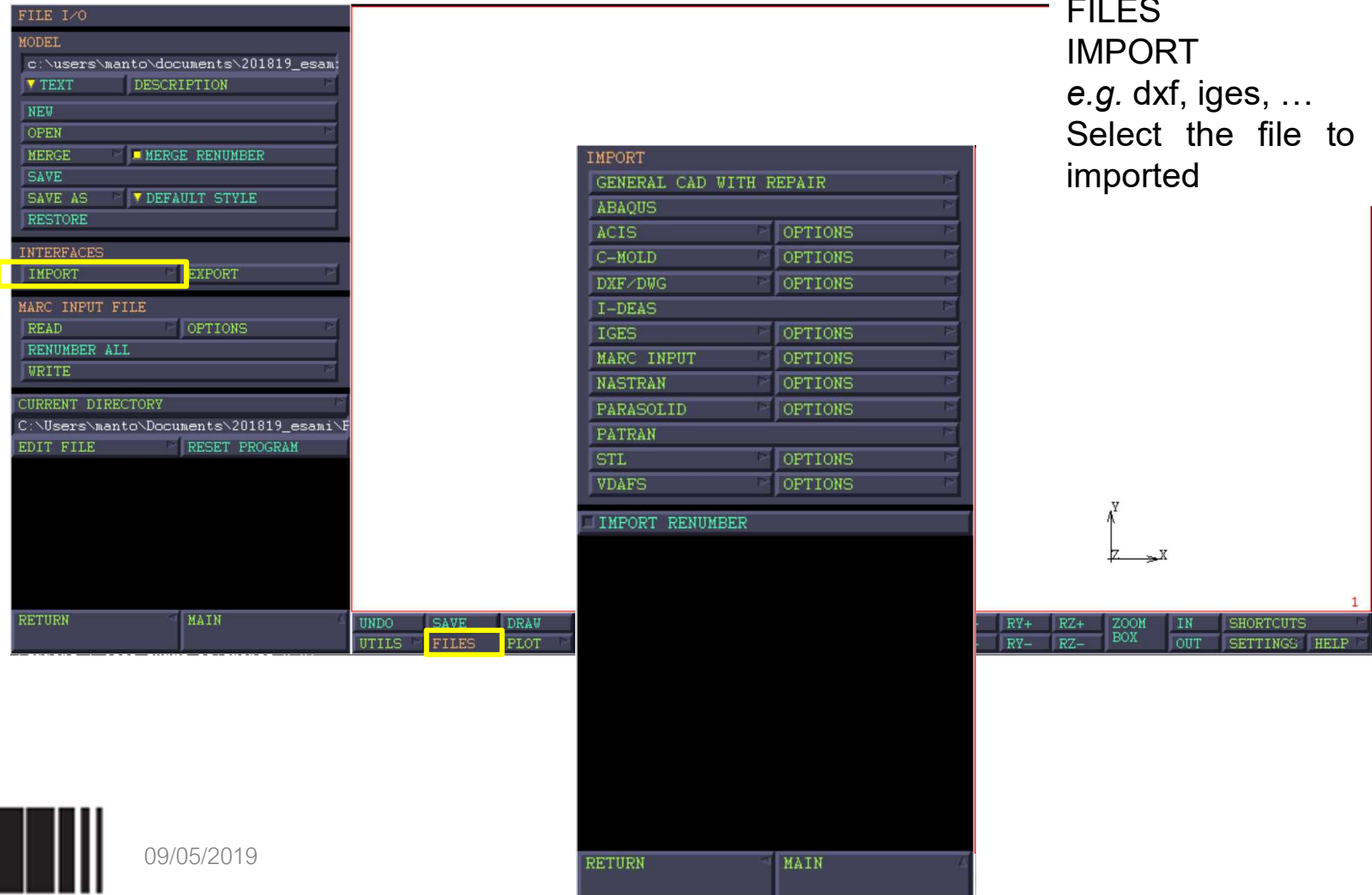
MAIN MENU

FILES

IMPORT

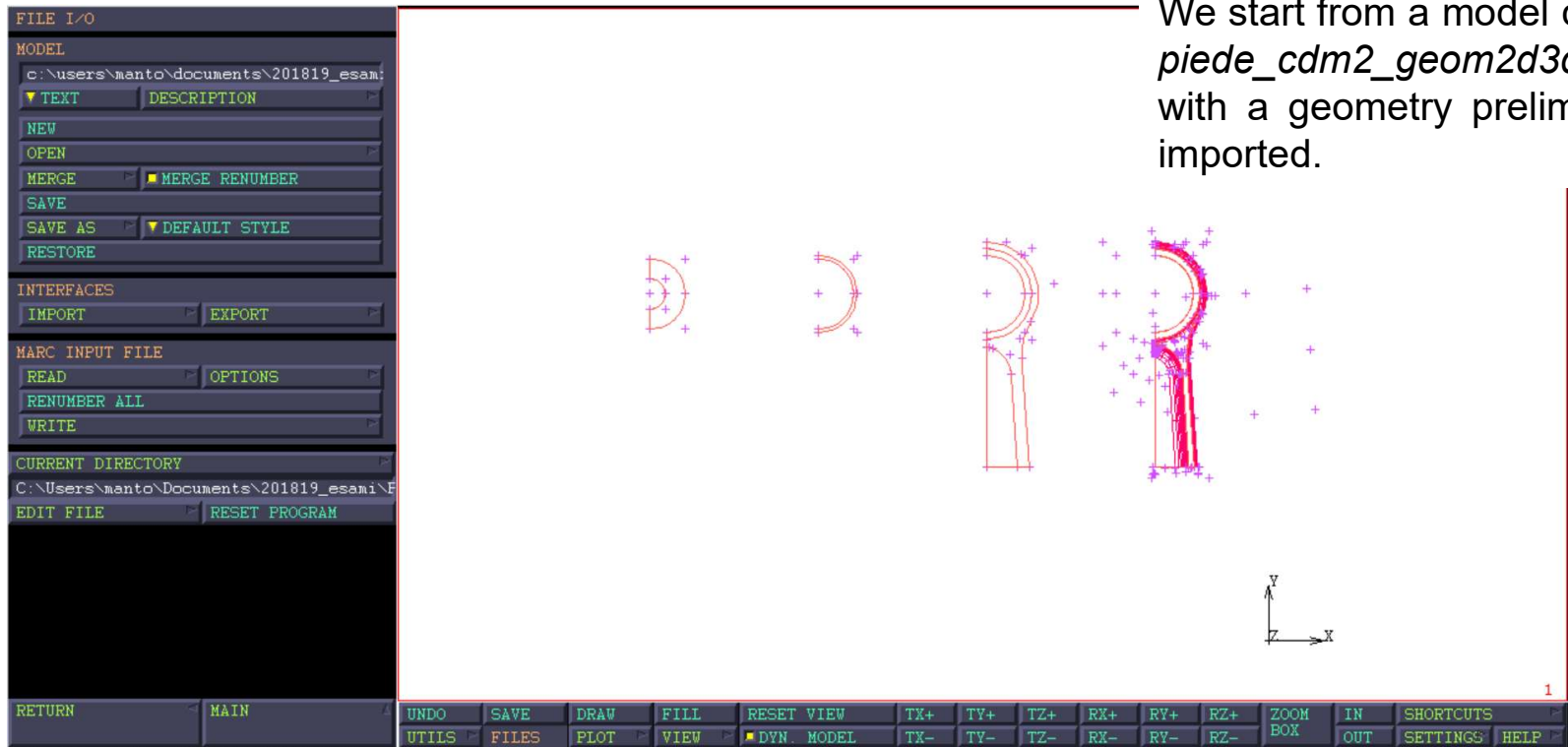
e.g. dxf, iges, ...

Select the file to be imported



# The geometry import

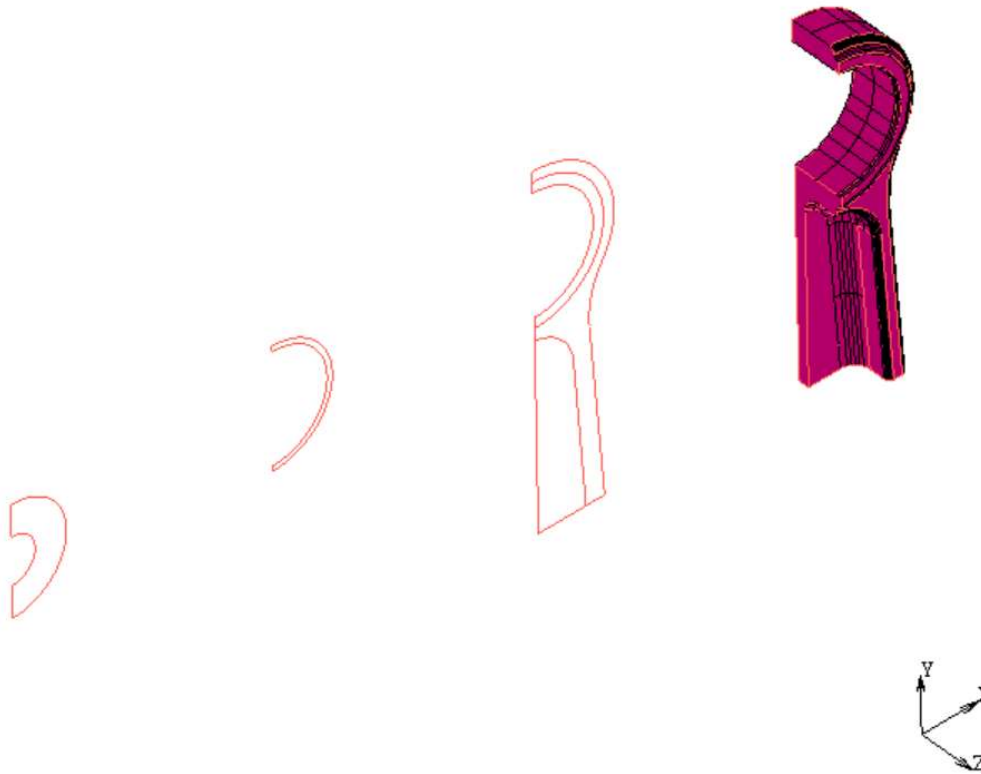
## Connecting rod and bush: interference loadcase



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:

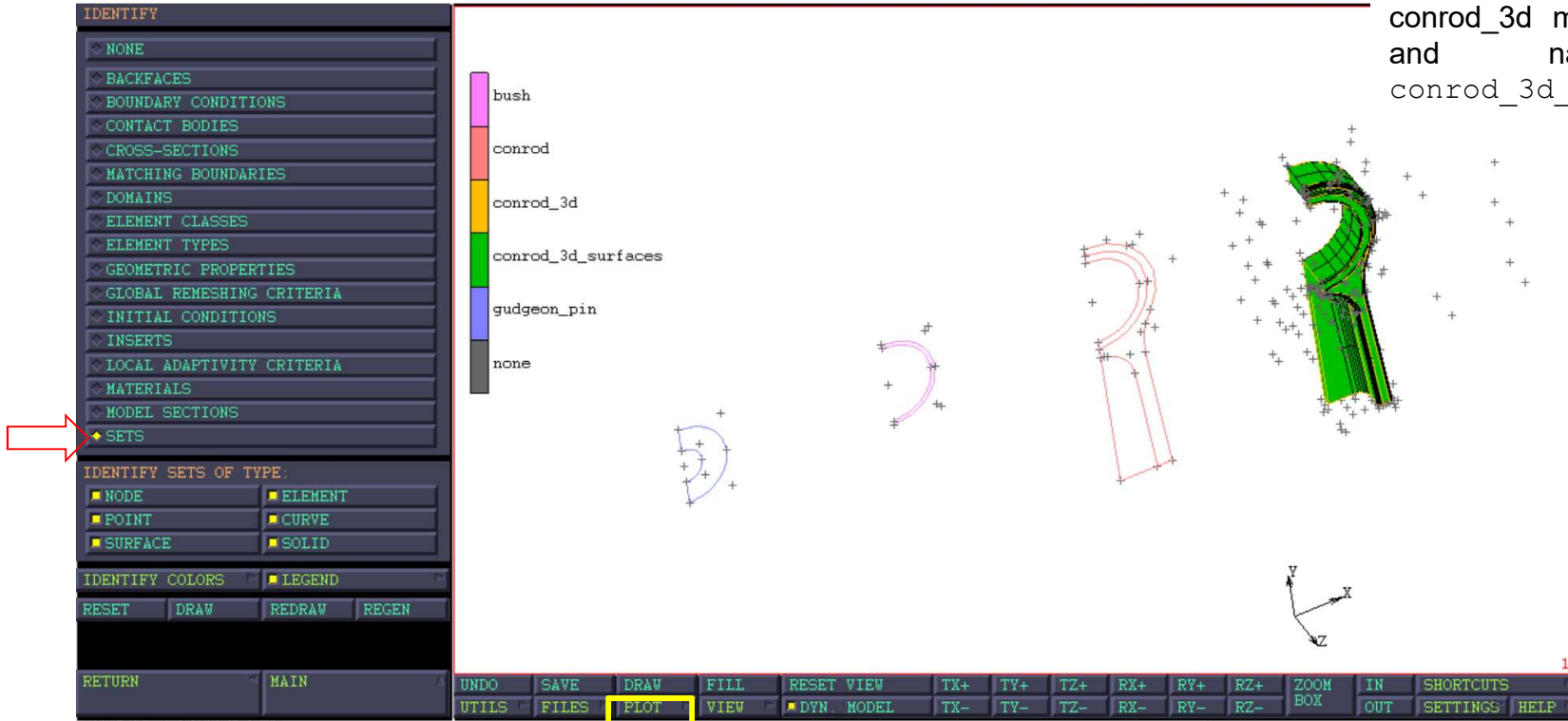
- Connecting rod 3D;
- Connecting rod 2D;
- Bush;
- Gudgeon pin.

# 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.

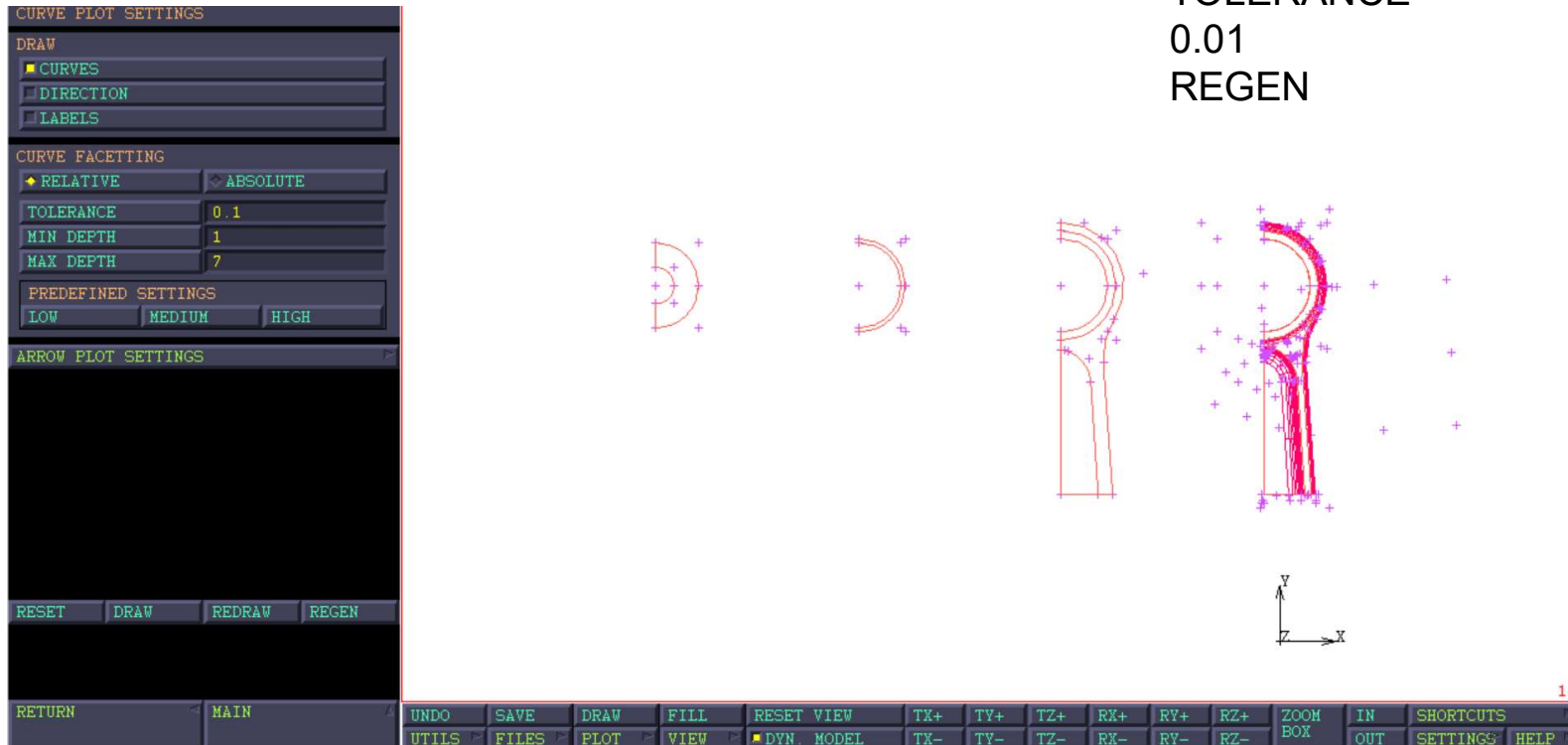




# The geometry import

## 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**

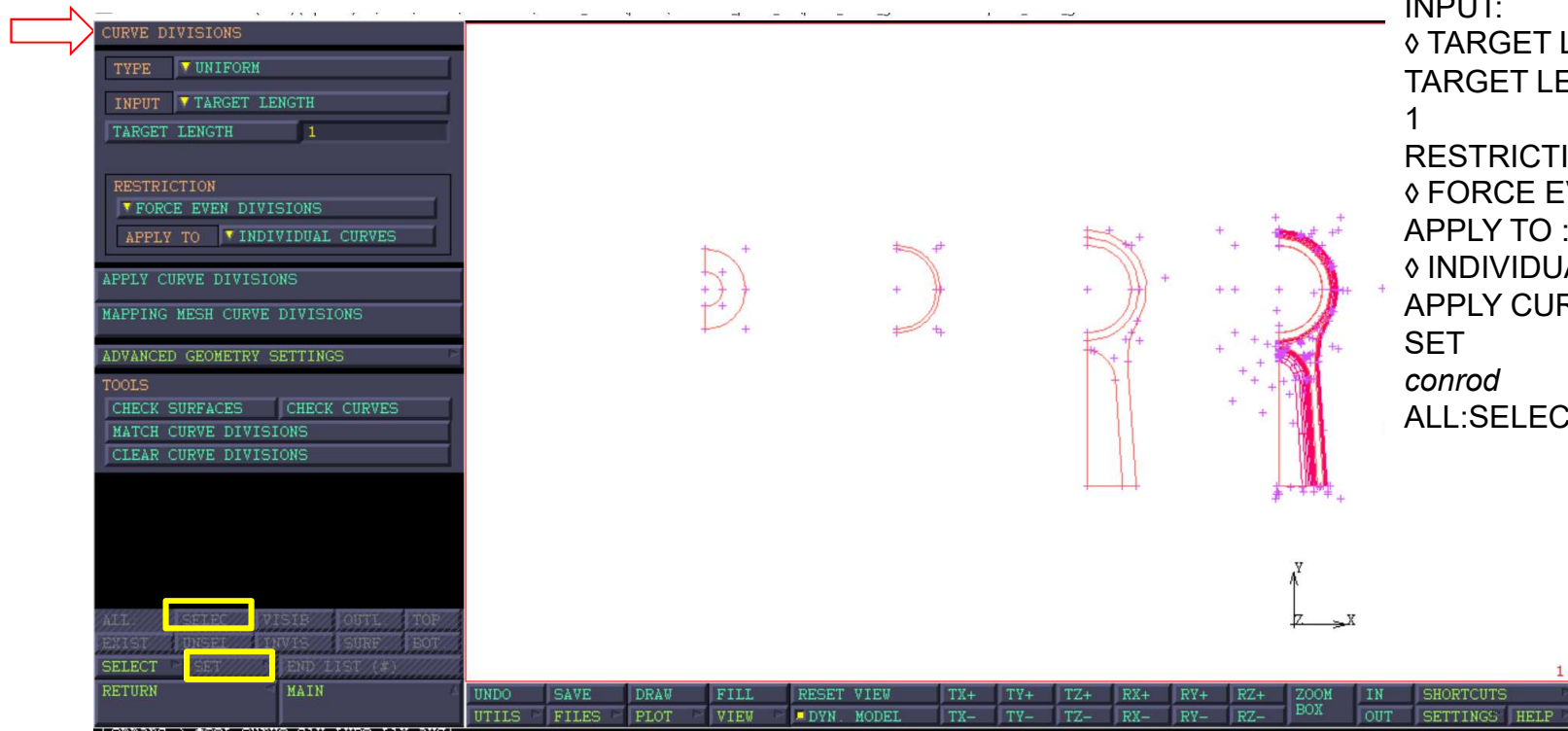
Contact

References

# Mesh generation

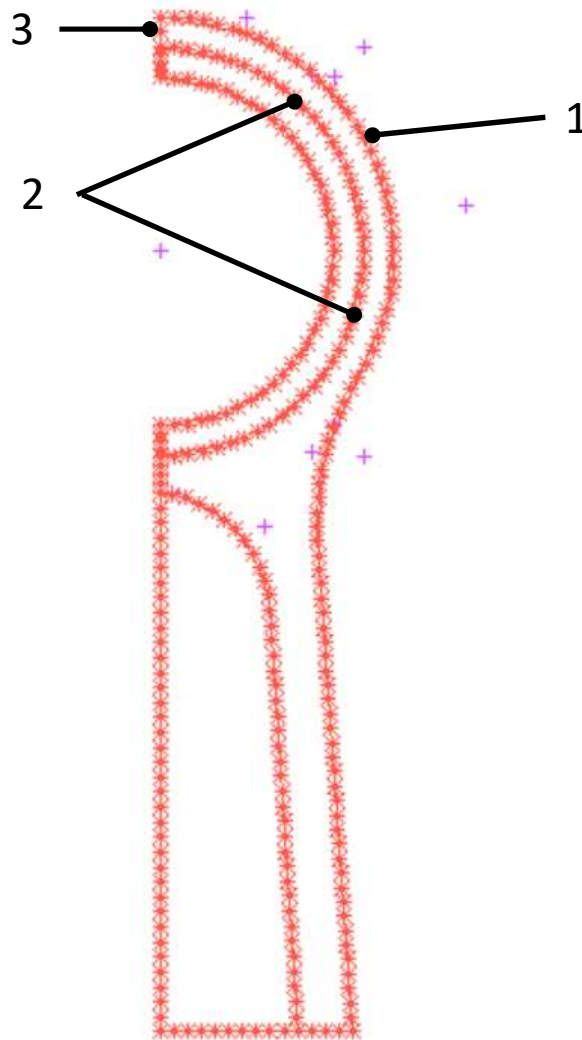
## Connecting rod 2D model

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



1 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 circumferential edge at small end;
2. Middle circumferential 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.

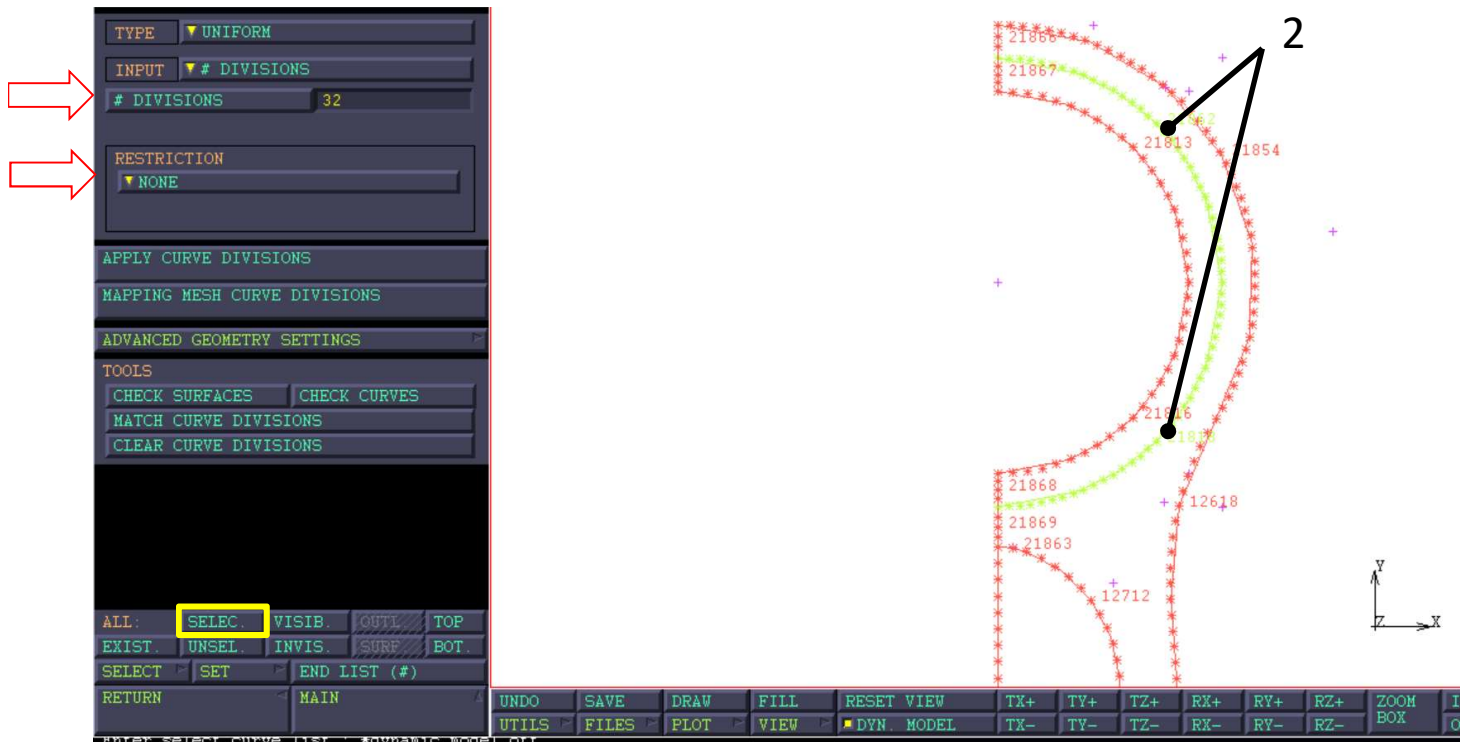
The screenshot displays the 'CURVE DIVISIONS' dialog box on the left and a 2D model of a connecting rod on the right. The dialog box settings are as follows:

- TYPE: UNIFORM
- INPUT: # DIVISIONS
- # DIVISIONS: 43
- RESTRICTION: NONE
- APPLY TO: INDIVIDUAL CURVES

The 'SELECT' button in the dialog box is highlighted with a yellow box. The 2D model shows a connecting rod with a mesh of red and green points. A black arrow points to a specific curve labeled '21854'.

# 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 small end at the middle 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.

CURVE DIVISIONS  
 TYPE ▾ UNIFORM  
 INPUT ▾ # DIVISIONS  
 # DIVISIONS 3  
 RESTRICTION  
 ▾ NONE  
 APPLY CURVE DIVISIONS  
 MAPPING MESH CURVE DIVISIONS  
 ADVANCED GEOMETRY SETTINGS  
 TOOLS  
 CHECK SURFACES CHECK CURVES  
 MATCH CURVE DIVISIONS  
 CLEAR CURVE DIVISIONS  
 ALL: SELEC. VISIB. OUTL TOP  
 EXIST. UNSEL. INVIS. SURE 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



# Mesh generation

## Connecting rod 2D model

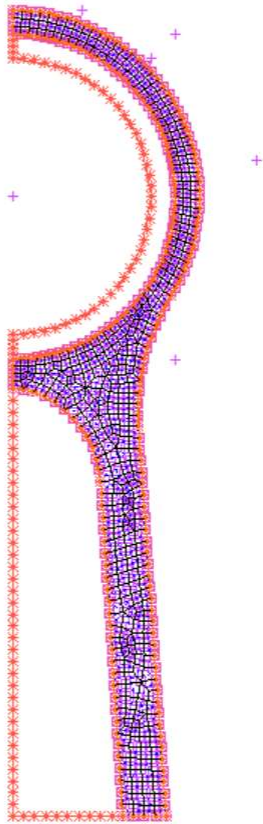
AUTOMESH 2-D PLANAR  
 MESH COARSENING PARAMETER  
 TRANSITION 1  
 QUADRILATERALS (ADV FRNT)  
 QUAD MESH!  
 QUADRILATERALS (OVERLAY)  
 DIVISIONS 10  
 BIAS FACTORS 0  
 QUAD MESH! ADVANCED  
 TRIANGLES (DELAUNAY)  
 TRI MESH!  
 TRIANGLES (ADV FRNT)  
 TRI MESH!  
 QUAD/TRI MIXED (ADV FRNT)  
 MAX QUAD DISTORTION 0.9  
 QUAD/TRI MESH!  
 TOOLS  
 CHECK MESH CLEAR MESH  
 ALL: SELEC. VISIB. OUTL TOP  
 EXIST UNSEL. INVIS. SURE 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

21857  
 21862  
 21813  
 21854  
 21823  
 21816  
 21810  
 21866  
 21869  
 21863  
 21871  
 21837  
 12712  
 21863  
 21869  
 21818  
 21862  
 21866  
 21870  
 21849  
 21837  
 21871  
 21871

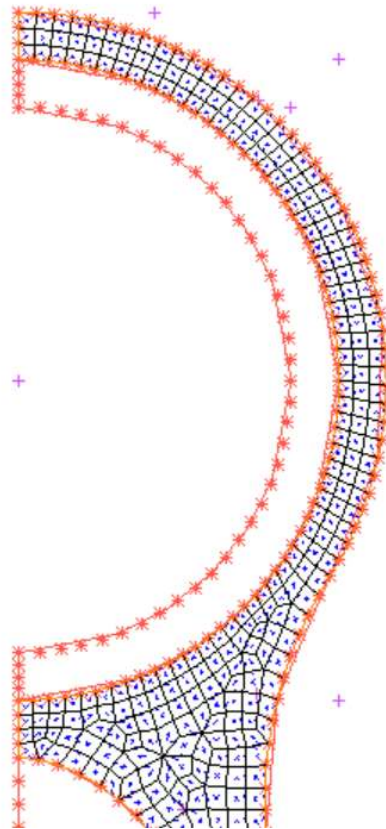
MAIN  
 MESH GENERATION  
 AUTOMESH  
 2-D PLANAR MESHING  
 QUADRILATERAL (ADV FRNT)  
 QUAD MESH!  
 Select manually the curves  
 shown beside with ID label:  
 21854  
 12618  
 21849  
 21871  
 21837  
 12712  
 21863  
 21869  
 21818  
 21862  
 21866  
 # | End of List  
 ALL:SELEC.

# Mesh generation

## Connecting rod 2D model



Global view



Detailed view

The mesh is uniformly 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

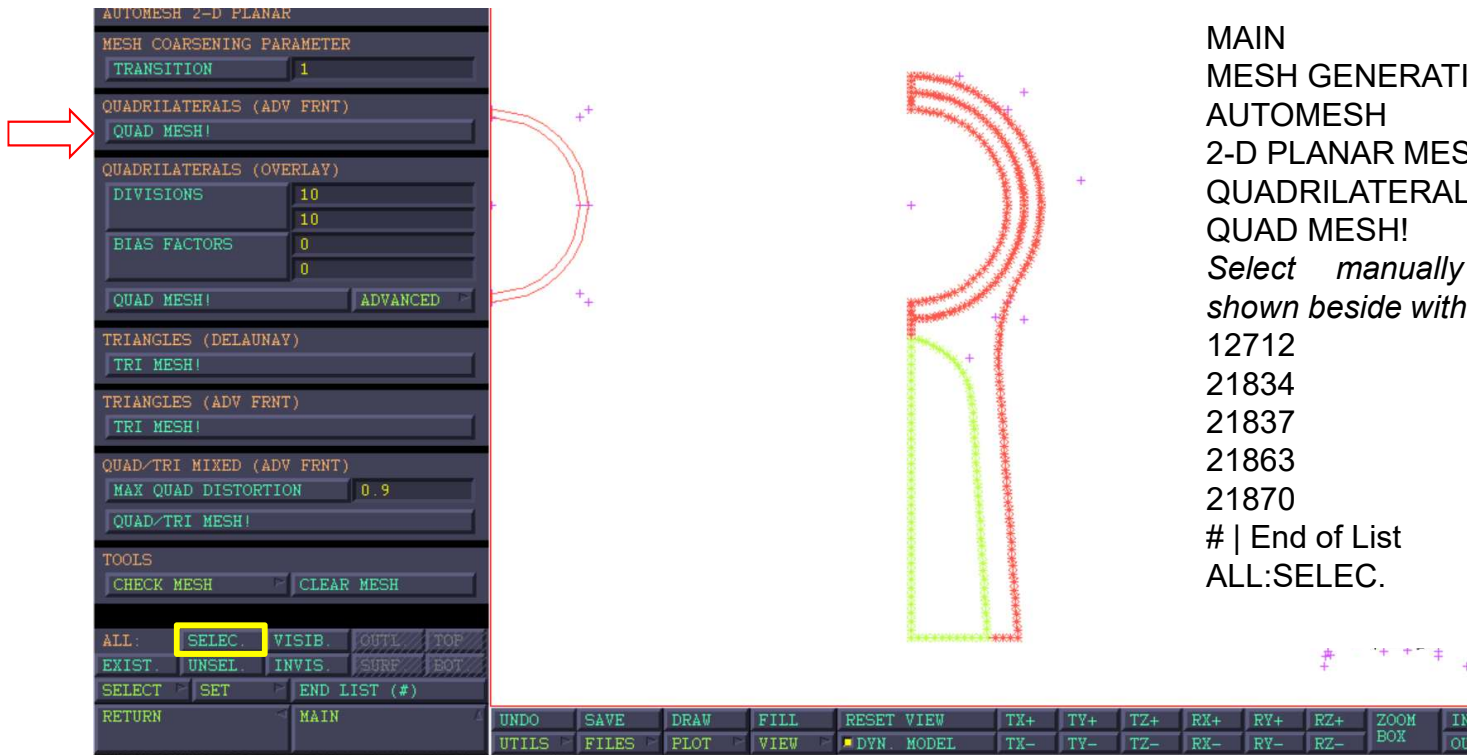
Allows the sets (e.g. elements, curves, ...) to be visible ( ALL) or invisible ( NONE) selectively.

NAME	TYPE	COUNT	VISIBLE	
bush	curve	4	<input checked="" type="checkbox"/> ALL <input type="checkbox"/> NONE	4
conrod	curve	17	<input checked="" type="checkbox"/> ALL <input type="checkbox"/> NONE	17
gudgeon_pin	curve	4	<input checked="" type="checkbox"/> ALL <input type="checkbox"/> NONE	4
shank_elems	element	403	<input type="checkbox"/> ALL <input checked="" type="checkbox"/> NONE	0



# Mesh generation

## Connecting rod 2D model

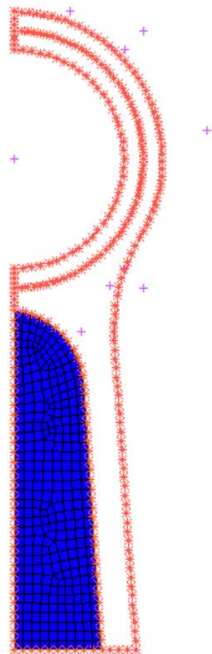


MAIN  
 MESH GENERATION  
 AUTOMESH  
 2-D PLANAR MESHING  
 QUADRILATERAL (ADV FRNT)  
 QUAD MESH!  
*Select manually the curves shown beside with ID label:*  
 12712  
 21834  
 21837  
 21863  
 21870  
 # | End of List  
 ALL:SELEC.

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_elems`.

# Mesh generation

## Connecting rod 2D model

SELECT

NODES	0	CLR	STORE
ELEMENTS	0	CLR	STORE
.. EDGES	0	CLR	STORE
.. FACES	0	CLR	STORE
POINTS	0	CLR	STORE
CURVES	0	CLR	STORE
SURFACES	0	CLR	STORE
SOLIDS	0	CLR	STORE
VRTCS	0	CLR	STORE
EDGES	0	CLR	STORE
FACES	0	CLR	STORE

SELECT SET    SELECT BY

SELECT CONTACT BODY ENTITIES

FILTER    ▼ NONE

METHOD    ▼ SINGLE

SELECT MODE    ▼ AND

CLEAR SELECT    RESET

MAKE VISIBLE    MAKE INVISIBLE

■ EXCLUDE INVISIBLE BODIES

STORE NODE PATH    STORE ORDERED

SETS    RENAME    REMOVE    MERGE

DEL ENTRIES    **VISIBILITY**

■ IDENTIFY SETS    IDENTIFY

ALL    SELEC    VISIB    GUTL    TOP

EXIST    UNSEL    INVIS    SURF    BOT

SELECT    SET    END LIST (#)

RETURN    MAIN

bush

conrod

gudgeon\_pin

pocket\_elems

shank\_elems

none

To identify sets:

PLOT

SELECT

IDENTIFY SETS

VISIBILITY

SHANK\_ELEMS  ALL

POCKET\_ELEMS  ALL

OK

1

# 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.

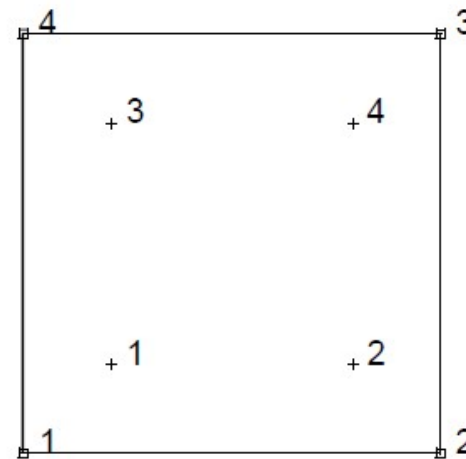
In general, one needs more of these lower-order elements than the higher-order elements such as 26 or 53. Hence, use a fine mesh.



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.

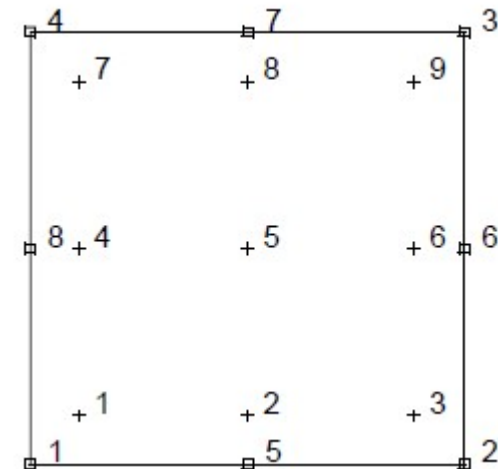
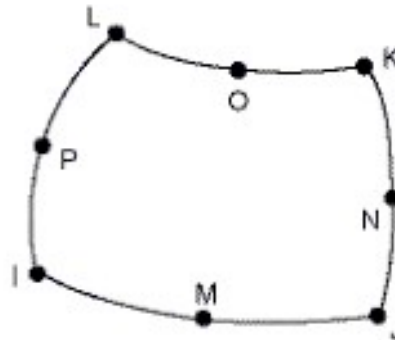


Lower-order elements, such as type 3, are preferred in contact analyses.

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.

The screenshot shows the 'MESH GENERATION' menu on the left, with 'CHANGE CLASS' highlighted in yellow. A 3D model of a part is shown in the center, with a mesh overlaid. On the right, the 'CHANGE CLASS' dialog is open, showing a list of element classes. The 'TO CLASS' section has 'QUAD (8)' selected with a yellow diamond. The 'ELEMENTS' section has 'EXIST.' highlighted in yellow. Red arrows indicate the flow from the menu to the dialog and then to the specific options.

# 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.

# Mesh generation

## Connecting rod 2D model – small end mesh

The screenshot displays the 'MESH GENERATION' panel on the left and a 2D model of a connecting rod on the right. The panel includes various options for nodes, elements, curves, surfaces, and solids, as well as settings for element class, curve type, surface type, and solid type. The 2D model shows a curved section of the rod with a dense mesh of quadrilateral elements at the small end, transitioning to a coarser mesh towards the larger end. A red arrow points to the 'SUBDIVIDE' option in the panel, and another red arrow points to the refined mesh area in the model. A text box in the center reads: 'The 1D-element line (3) to be subdivided.' The interface also features a coordinate system, a toolbar with various functions like 'CLEAR MESH', 'ATTACH', and 'SUBDIVIDE', and a status bar at the bottom.

# 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

SELECT			
NODES	0	CLR	STORE
ELEMENTS	0	CLR	STORE
... EDGES	0	CLR	STORE
... FACES	0	CLR	STORE
POINTS	0	CLR	STORE
CURVES	0	CLR	STORE
SURFACES	0	CLR	STORE
SOLIDS	0	CLR	STORE
VERTCS	0	CLR	STORE
EDGES	0	CLR	STORE
FACES	0	CLR	STORE
SELECT SET	SELECT BY		
SELECT CONTACT BODY ENTITIES			
FILTER	▼ NONE		
METHOD	▼ SINGLE		
SELECT MODE	▼ AND		
CLEAR SELECT	RESET		
MAKE VISIBLE	MAKE INVISIBLE		
■ EXCLUDE INVISIBLE BODIES			
STORE NODE PATH	STORE ORDERED		
SETS	RENAME	REMOVE	MERGE
DEL ENTRIES	VISIBILITY		
IDENTIFY SETS IDENTIFY			
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

# Mesh generation

## Connecting rod 2D model – small end mesh

SELECT BY  
NODES BY 0 CLEAR  
ELEMS EDGES FACES PTS  
CRVS SRFS TRANSFORMATION  
ELEMENTS BY 0 CLEAR  
NODES EDGES FACES TVPE  
**CLASS** GEOMETRY MATERIAL  
ORIENTATION CONTACT BODY  
EDGES BY 0 CLEAR  
NODES ELEMS FACES CRVS  
SRFS  
FACES BY 0 CLEAR  
NODES ELEMS EDGES SRFS  
POINTS BY 0 CLEAR  
CRVS SRFS SRFS (TR PTS)  
NODES  
CURVES BY 0 CLEAR  
PTS SRFS (TR CRVS) NODES  
EDGES GEOMETRY CONTACT BODY  
SURFACES BY 0 CLEAR  
PTS TR PTS TR CRVS NODES EDGES  
FACES GEOMETRY CONTACT BODY  
♦ ANY IN LIST ◀ ALL IN LIST  
SELECT MODE ▼ AND  
CLEAR SELECT RESET  
ALL SELEC VISIB GUTL TOP  
EXIST UNSEL INVIS SURF BOT  
SELECT SET END LIST (#)  
RETURN MAIN

SELECT ELEMENTS BY CLASS  
LINE (2) LINE (3)  
TRIA (3) TRIA (6)  
QUAD (4) QUAD (6)  
QUAD (8) QUAD (9)  
TETRA (4) TETRA (10)  
PENTA (6) PENTA (15)  
HEX (8) HEX (12)  
OK

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

1

# Mesh generation

## Connecting rod 2D model – small end mesh

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

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

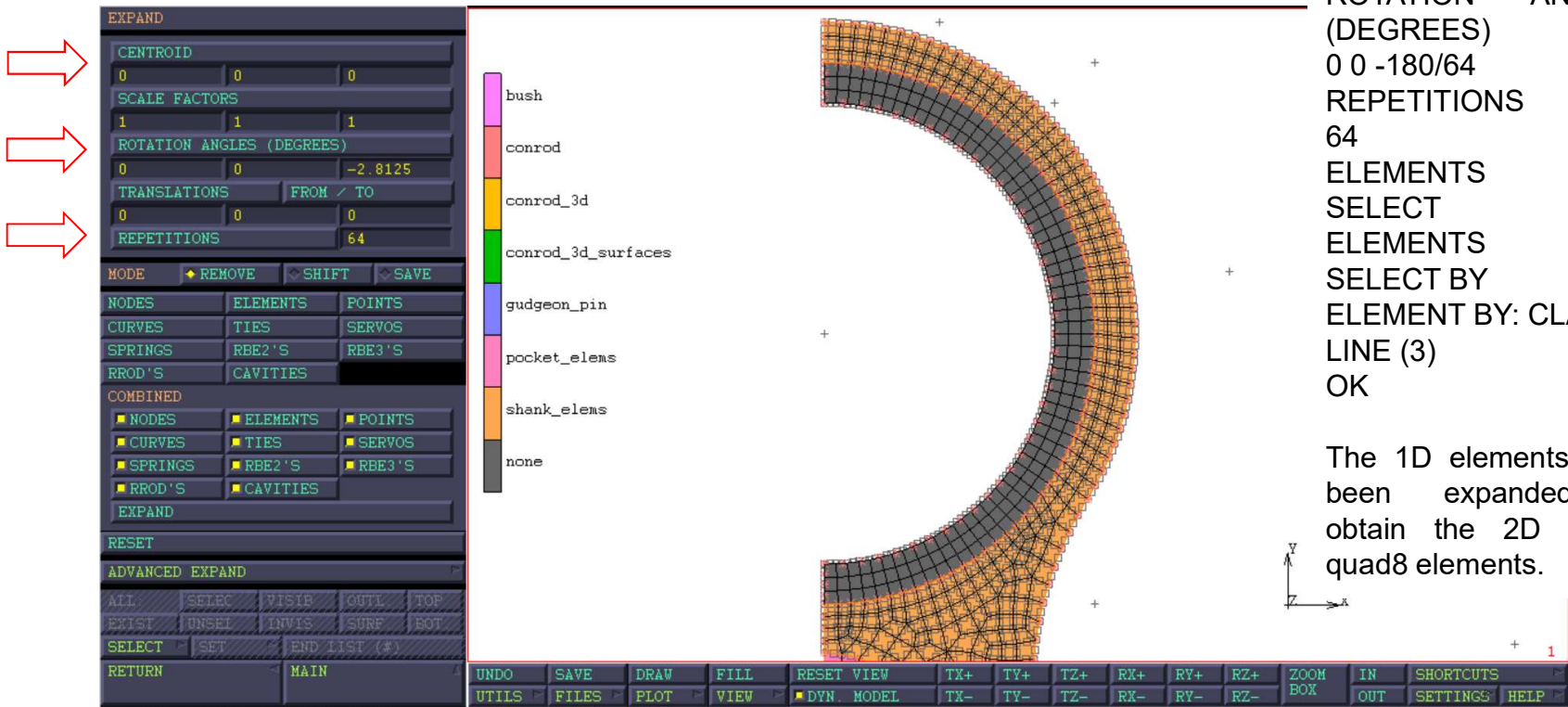


# Mesh generation

## Connecting rod 2D model – small end mesh

MAIN  
MESH GENERATION  
EXPAND  
CENTROID  
0 0 0  
ROTATION ANGLES  
(DEGREES)  
0 0 -2.8125  
TRANSLATIONS FROM / TO  
0 0 0  
REPETITIONS 64  
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  
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

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



# Mesh generation

## Connecting rod 2D model – small end mesh

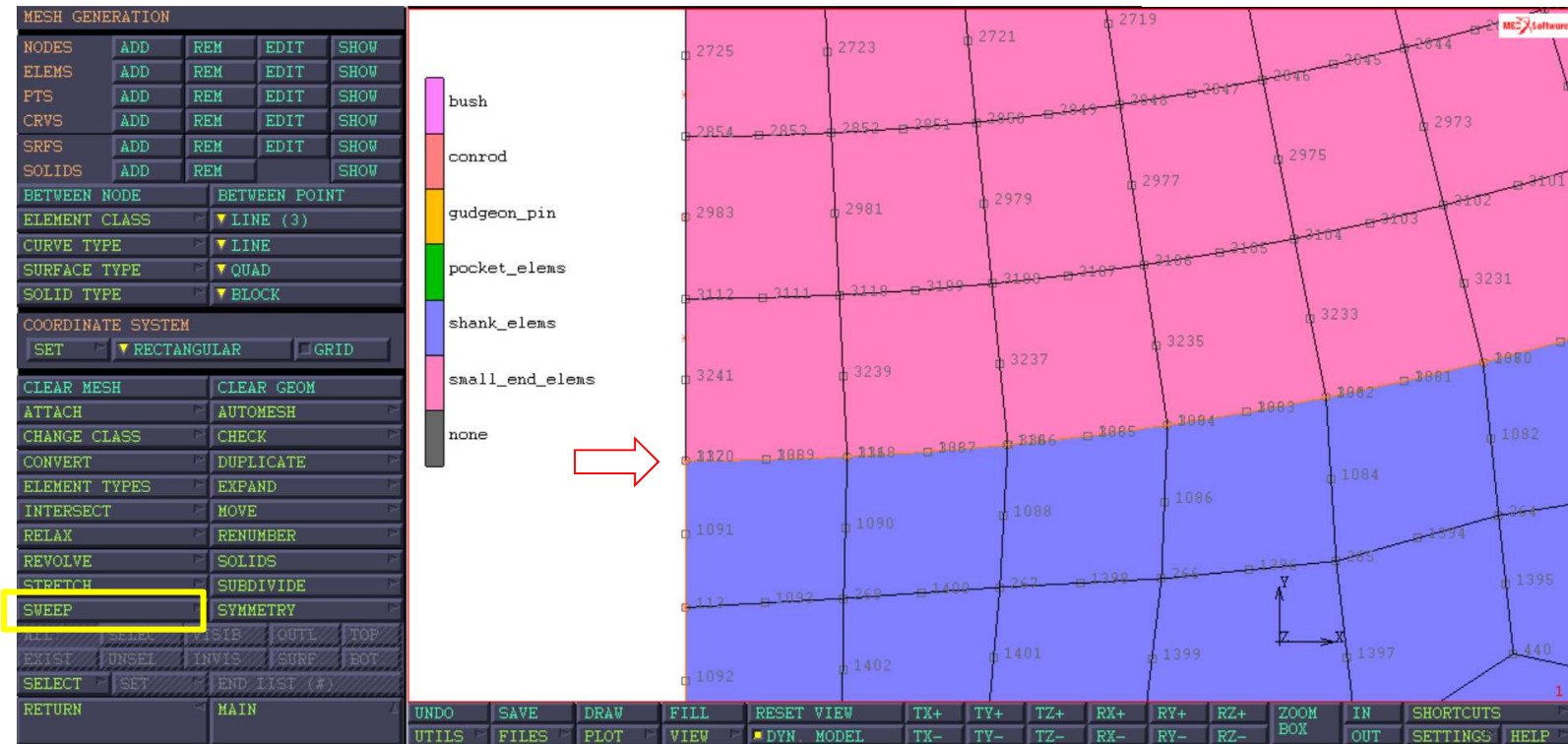
```
|-----  
*visible_all_sets  
*identify_sets *regen  
*invisible_set shank_elems  
*invisible_set pocket_elems  
*select_clear_elements  
*select_elements  
all_visible  
*store_elements small_end_elems  
all_selected  
*select_clear_elements  
*visible_all_sets  
|-----
```

At the end, these elements are stored, and the set is called small\_end\_elems

# Mesh generation Sweep

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

MAIN  
MESH GENERATION  
SWEEP



# Mesh generation

## Sweep

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

MAIN  
MESH GENERATION  
SWEEP  
Tolerance  
0.05  
NODES  
ALL: EXISTS.

In the command prompt, you read:  
Deleting 219 duplicate nodes!  
Deleting 0 collapsed elements!

```

Command > *dynamic_model_on
Command > *sweep_nodes
Enter sweep node list : all_existing
Deleting 219 duplicate nodes !
Deleting 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				
NODES	ADD	REM	EDIT	SHOW
ELEMS	ADD	REM	EDIT	SHOW
PTS	ADD	REM	EDIT	SHOW
CRVS	ADD	REM	EDIT	SHOW
SRFS	ADD	REM	EDIT	SHOW
SOLIDS	ADD	REM		SHOW
BETWEEN NODE		BETWEEN POINT		
ELEMENT CLASS	▼ QUAD (4)			
CURVE TYPE	▼ LINE			
SURFACE TYPE	▼ QUAD			
SOLID TYPE	▼ BLOCK			
COORDINATE SYSTEM				
SET	▼ 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			

# Mesh generation

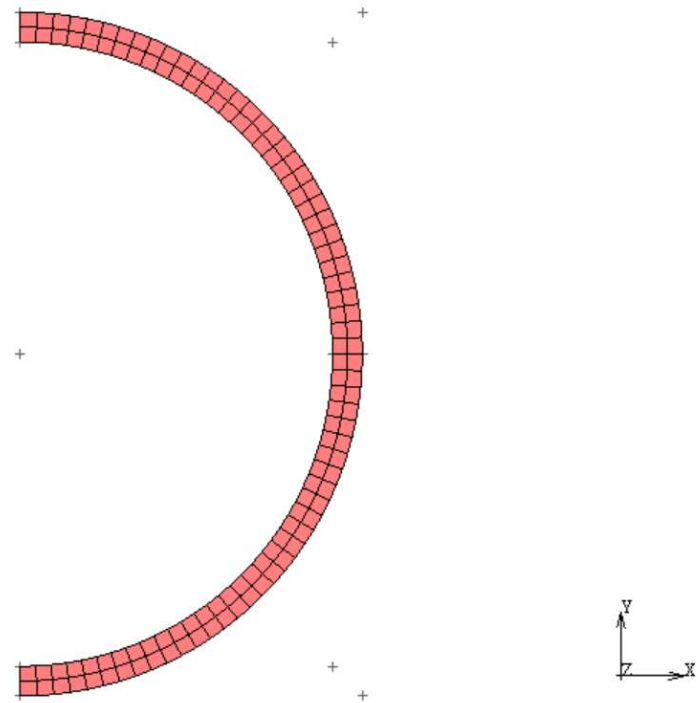
## Bush

```
-----  
|   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

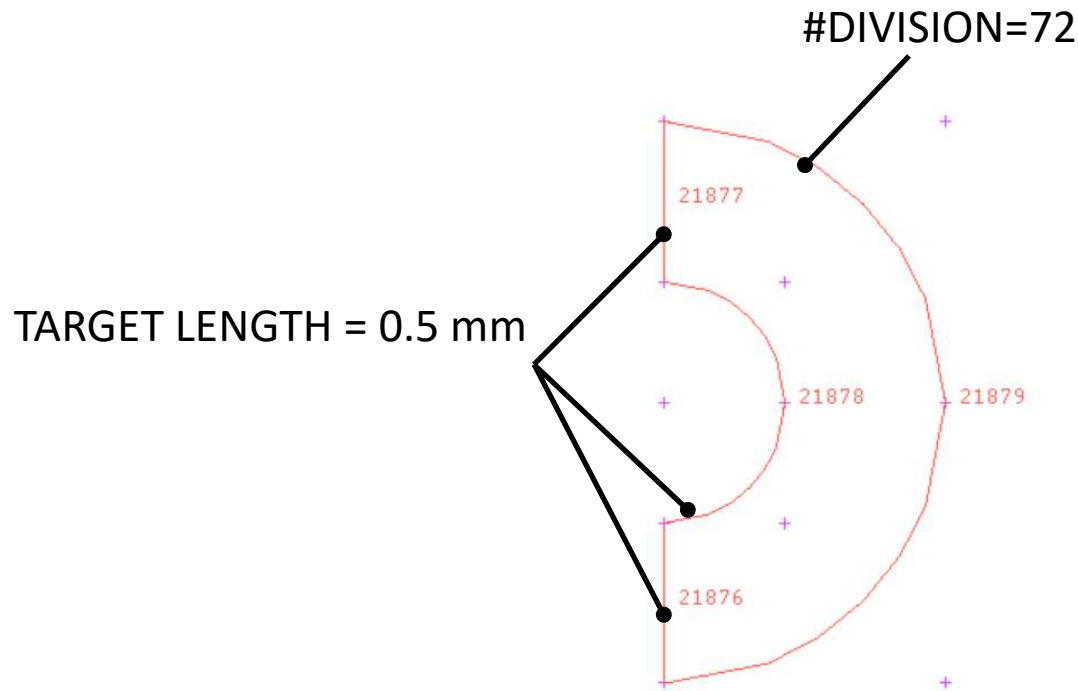
```
-----  
| MESH GENERATION: expand  
| From 1D quadratic elements (line3)  
| to 2D-planar quadratic elements (quad8).  
-----  
*expand_reset  
*set_expand_point  
-50  
0  
0  
*set_expand_repetitions 64  
*set_expand_rotation z -180/64  
*expand_elements  
all_selected  
-----  
| STORE and PLOT 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.



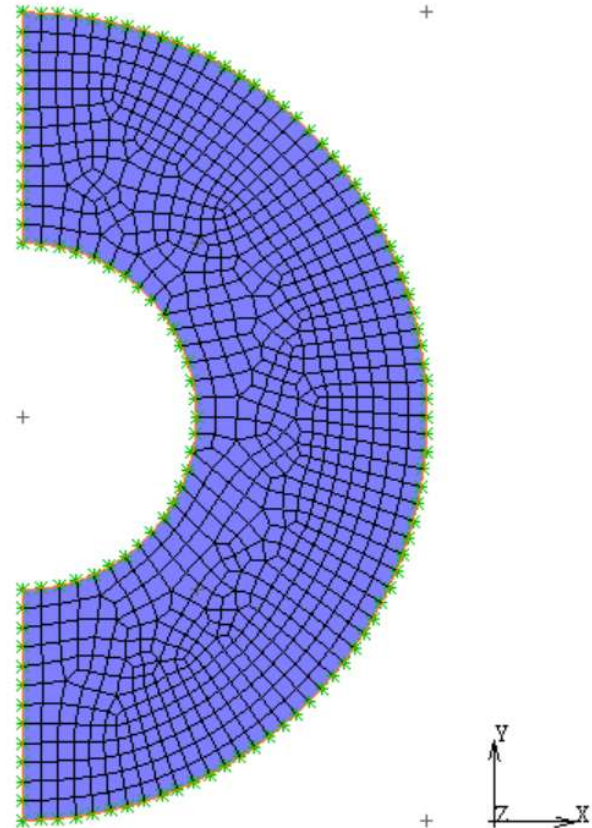
```
-----  
| Outer gudgeon pin curve  
-----  
*select_clear_curves  
*set_curve_div_type_fix_ndiv  
*set_curve_div_num  
72  
*set_curve_div_rest_off  
*apply_curve_divisions  
21879  
# | End of List  
*select_clear_curves  
-----  
| Inner and vertical gudgeon pin curves  
-----  
*set_curve_div_type_fix  
*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

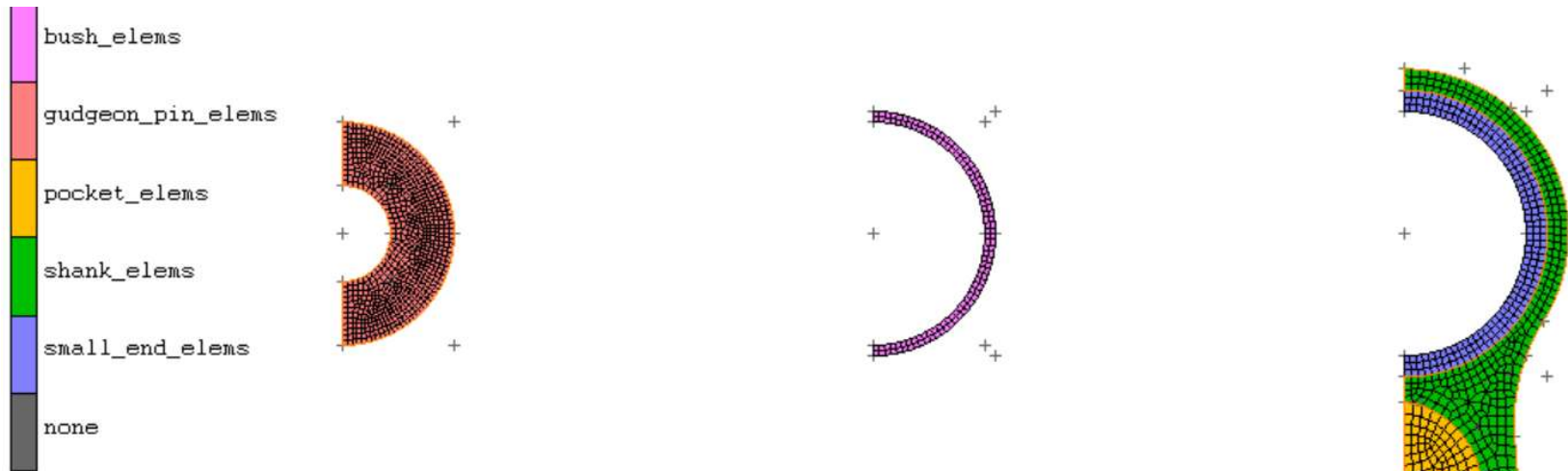
## Gudgeon pin

```
-----  
| 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 ( $\epsilon_z = 0, \gamma_{xz} = 0, \gamma_{yz} = 0$ ).

MAIN  
 GEOMETRIC PROPERTIES  
 NEW  
 STRUCTURAL  
 PLANAR  
 PLANE STRESS  
 NAME  
 Type e.g. shank\_elems\_18mm

# Mesh generation

## The geometric properties

GEOMETRIC PROPERTIES  
ANALYSIS CLASS  
STRUCTURAL  
NEW REM  
NAME shank\_elems\_18mm  
TYPE mech\_planar\_pstress  
COPY PREV NEXT EDIT  
PROPERTIES  
BEAM SECTIONS TABLES  
PLOT SETTINGS  
BEAM SHELL  
THICKNESS DIRECTION  
ID GEOMETRIES TOOLS  
ELEMENTS ADD REM 403  
ALL SELEC VISIB OUTL TOP  
EXIST UNSEL INVIS SURF ROT  
SELECT SET END LIST (#)  
RETURN MAIN

PLANE STRESS STRUCTURAL PROPERTIES  
NORMAL TO PLANE  
THICKNESS 18  
ELEMENT TECHNOLOGY  
ASSUMED STRAIN  
CONSTANT TEMPERATURE  
CLEAR OK

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

shank\_e

none

CURRENTLY DEFINED SETS		
bush	curve	4
bush_elems	element	128
geared	curve	17
gudgeon_pin	curve	4
gudgeon_pin_elems	element	801
pocket_elems	element	319
shank_elems	element	403
small_end_elems	element	192

OK

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 properly the thickness as shown beside.

# Mesh generation

## Geometric properties

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

MAIN  
GEOMETRIC PROPERTIES  
NEW  
STRUCTURAL  
PLANAR  
PLANE STRAIN  
NAME  
Type  
gudgeon\_pin\_elems\_60mm  
PROPERTIES  
THICKNESS: 60  
OK  
ELEMENTS: ADD

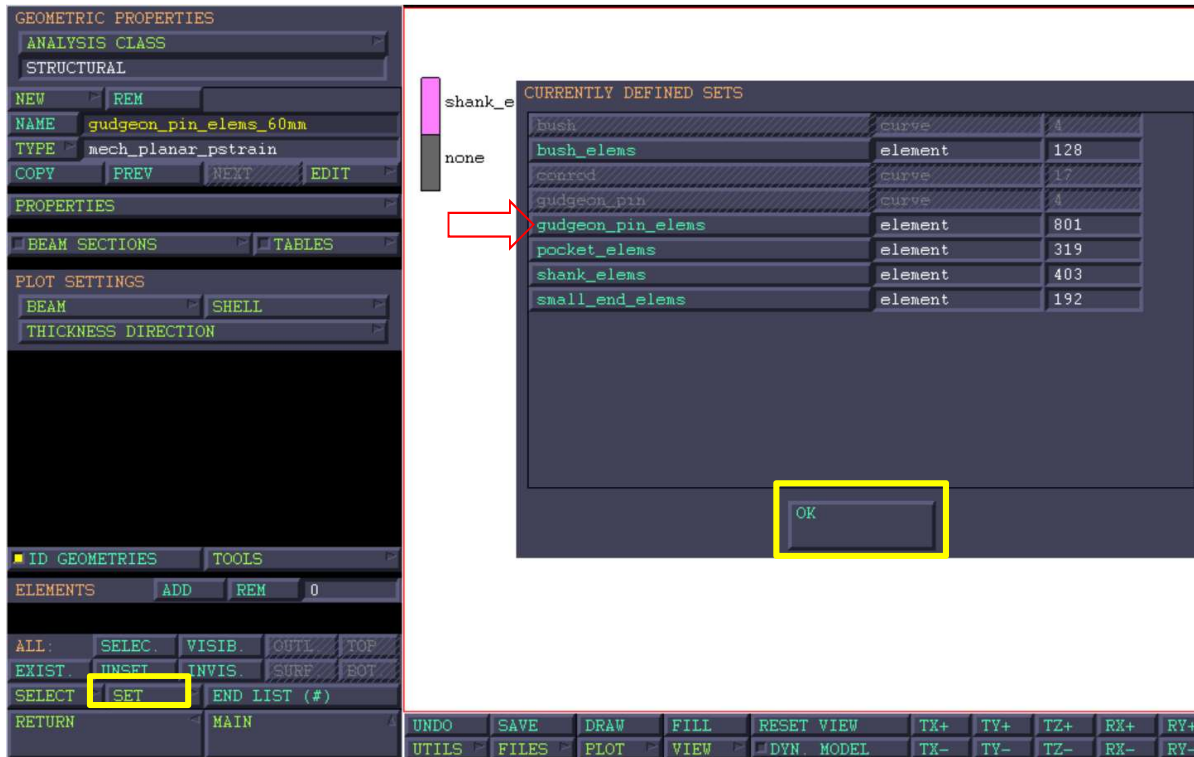
e.g.



# Mesh generation

## Geometric properties

Plane strain geometric property has been applied for the gudgeon 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.

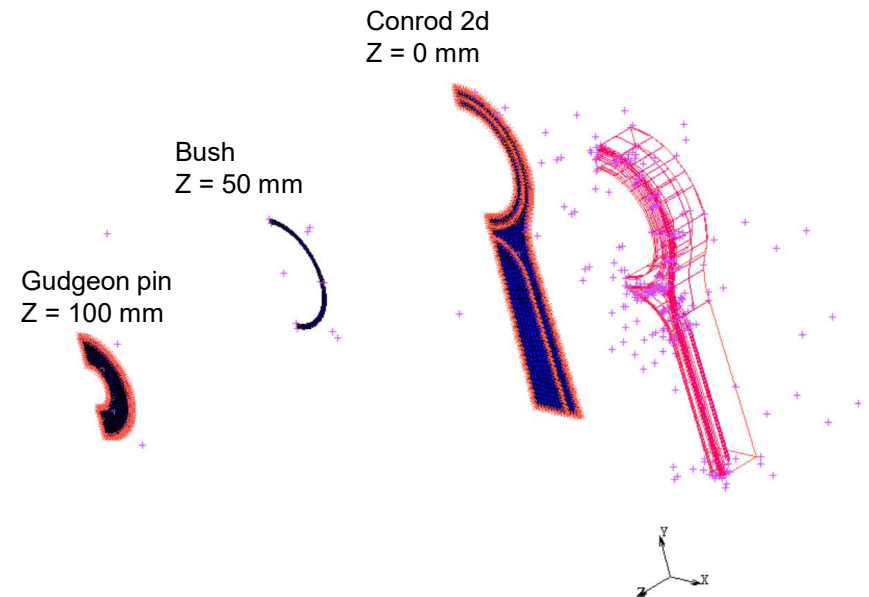
$E = 210000 \text{ MPa}$   
 $\nu = 0.3$

# 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

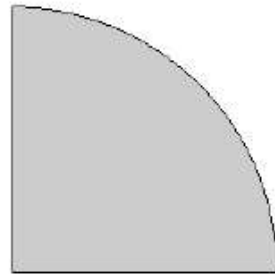
**Contact**

References

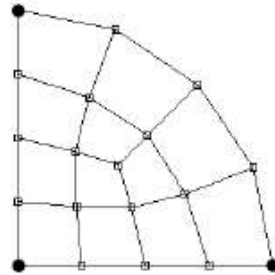
# Contact

## Contact bodies: ANALYTICAL vs DiSCRETE

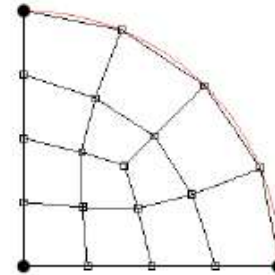
Planar Model



Actual Geometry



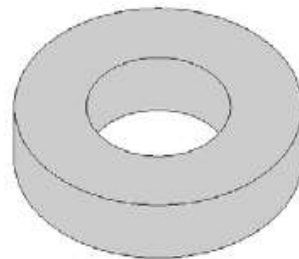
Finite Element Approximation



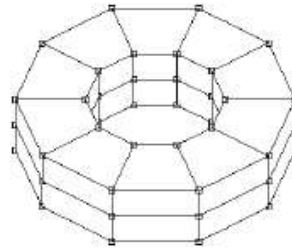
Cubic Spline Representation

●: Nodes with a normal vector discontinuity

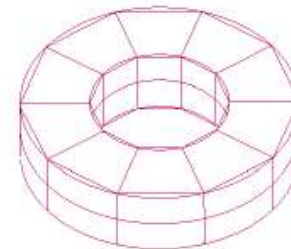
3D Model



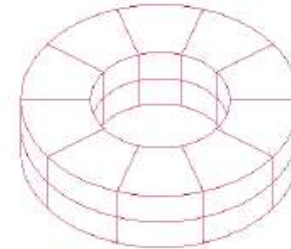
Actual Geometry



Finite Element Approximation



$C^0$ -discontinuous Coons  
Surface Representation



$C^0$ -continuous Coons  
Surface Representation

# 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.

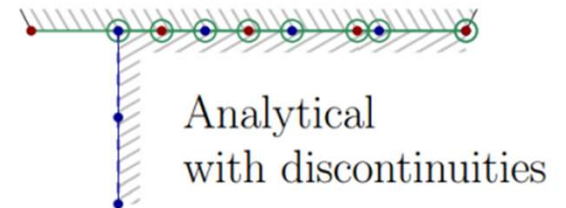
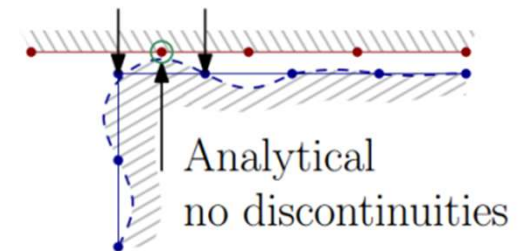
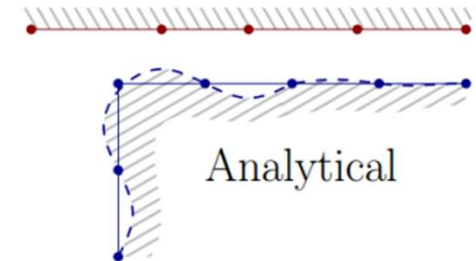
In the analytical description, linear segments are replaced by:

- 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.

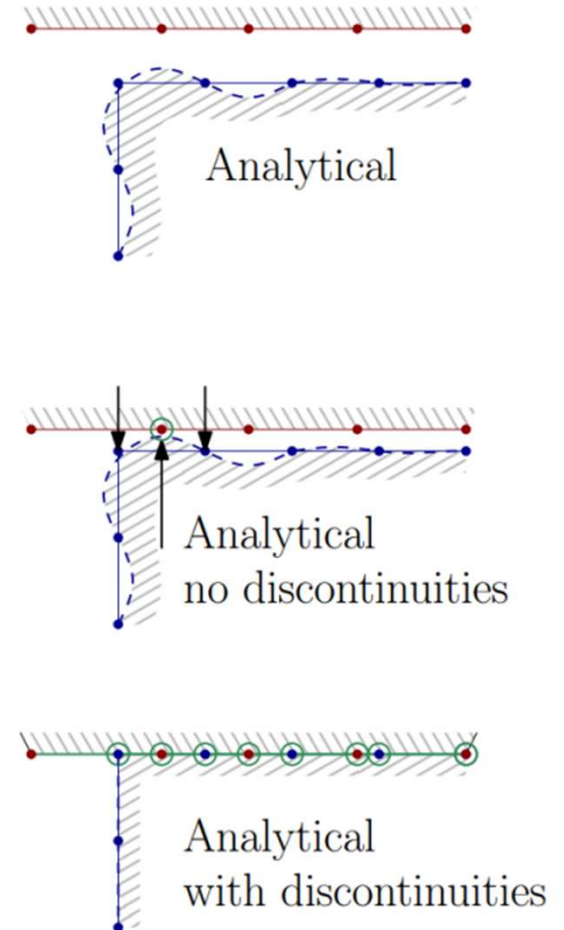


# Contact

## 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.

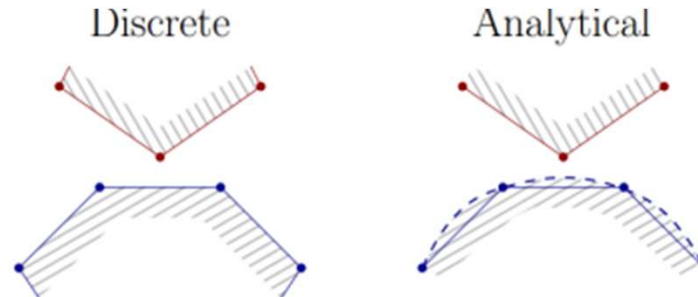




# Contact

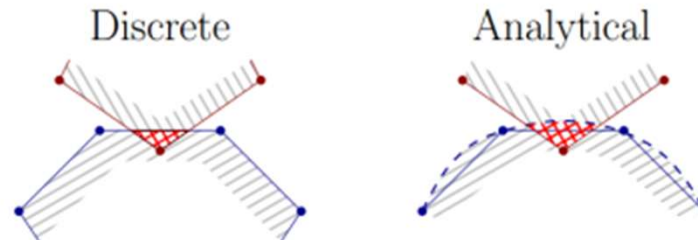
## Contact bodies

Time :  $t_1$   
Load:  $F = F_1$



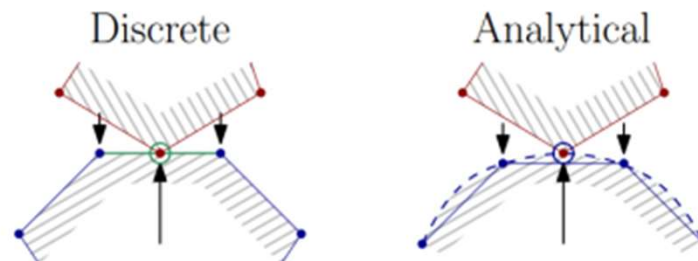
Contact CHECK:  
NO contact

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 :  $t_2$  (STEP 1)  
Load:  $F = F_2 > F_1$



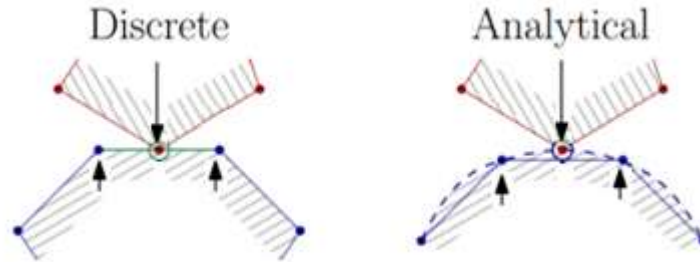
The force of the interaction  
between the bodies is a  
repulsive forces, that are  
balanced at the bodies nodes.

A kinematic relation between  
forces the nodes of the upper  
body to lie on the surface of  
the bottom body since .57

# Contact

## Contact bodies

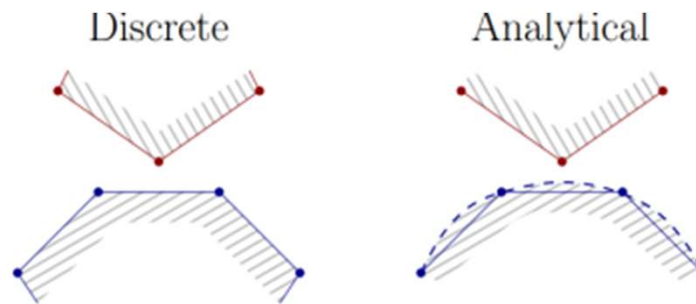
Time : t3 (STEP 0)  
 Load:  $F = F_1 < F_2$



CHECK CONTACT:

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

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



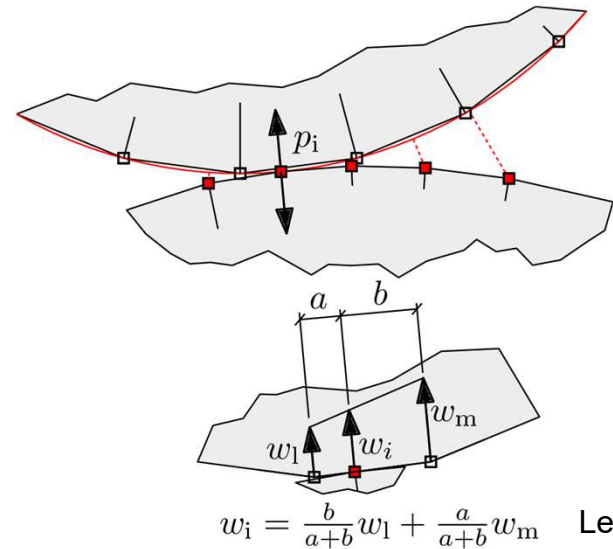
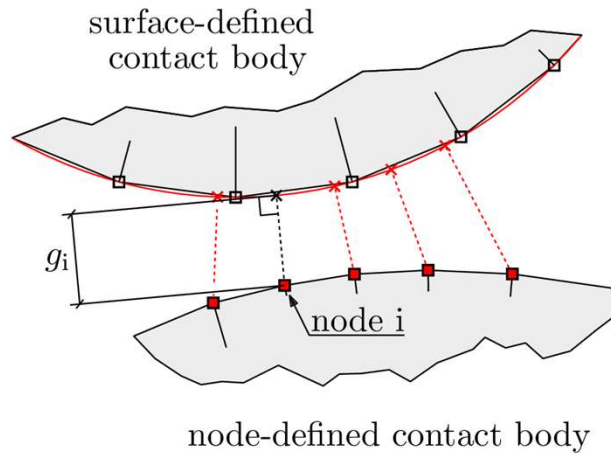
The Signorini inequalities relations that manage the unilateral contact definition cannot be solved in a FEM equation system; therefore, unilateral BCs are treated by activating or deactivating bilateral BCs.

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

# 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



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

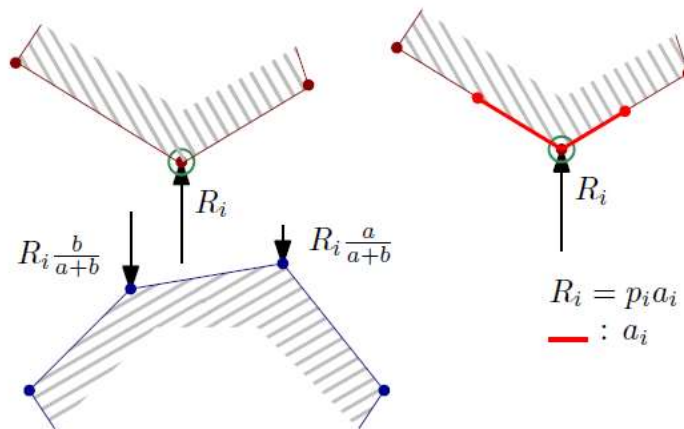
# Contact

## Contact bodies

The Signorini inequalities 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

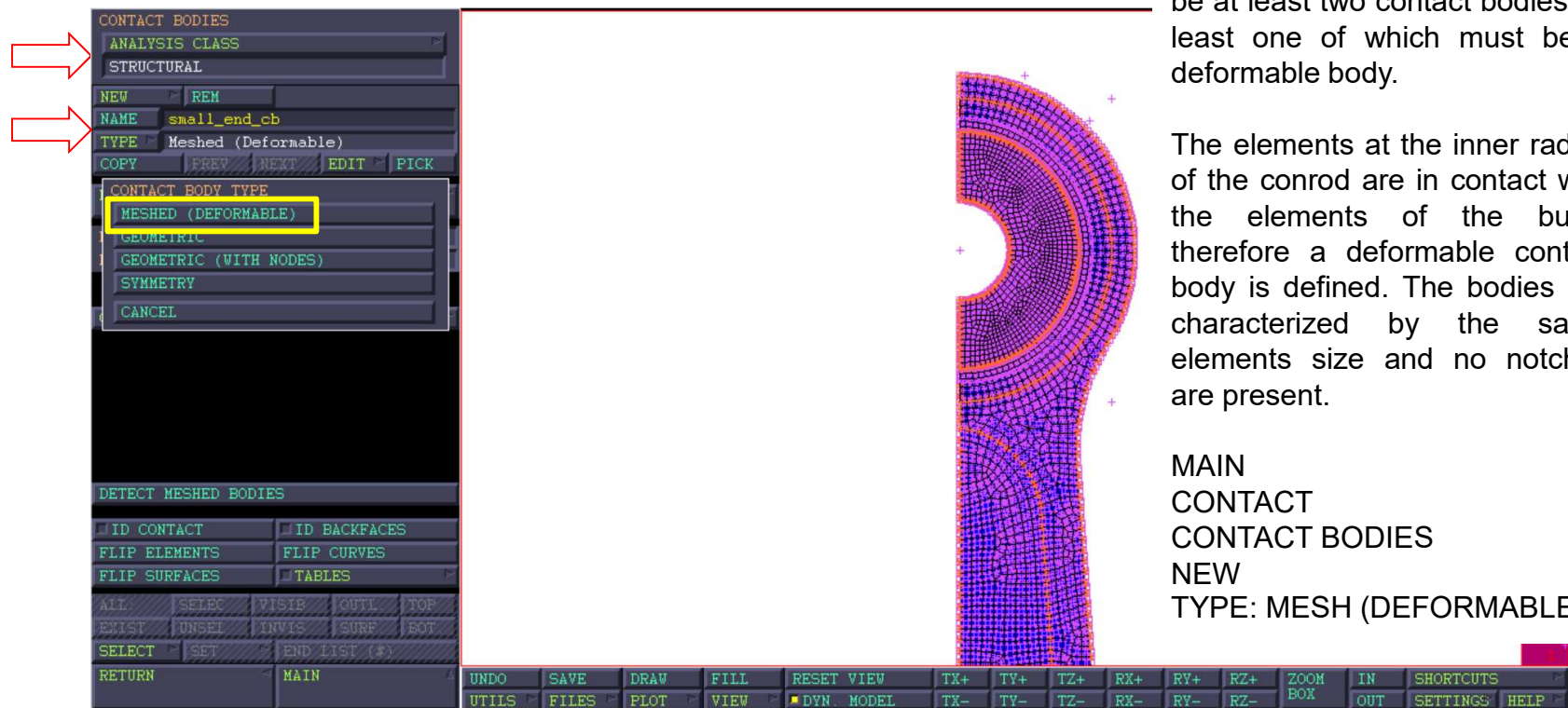
## Contact bodies: small\_end\_cb

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

The screenshot displays a software interface for defining contact bodies. On the left, the 'CONTACT BODIES' panel shows the 'NAME' as 'small\_end\_cb' and 'TYPE' as 'Meshed (Deformable)'. The 'PROPERTIES' section is highlighted with a yellow box. The 'MESHED BODY' panel on the right shows 'BOUNDARY DESCRIPTION' set to 'DISCRETE', also highlighted with a yellow box and a red arrow. The 'OK' button at the bottom right of the 'MESHED BODY' panel is highlighted with a yellow box. A red arrow points to the 'NEW' button in the 'CONTACT BODIES' panel. The background shows a 3D model of a meshed part with a coordinate system (X, Y, Z).

CONTACT BODIES  
ANALYSIS CLASS  
STRUCTURAL  
NEW REM  
NAME small\_end\_cb  
TYPE Meshed (Deformable)  
COPY PREV NEXT EDIT PICK  
PROPERTIES  
MODEL SECTIONS ADD REM 0  
ELEMENTS ADD REM 0  
CONTACT BODY VISIBILITY  
DETECT MESHED BODIES  
ID CONTACT ID BACKFACES  
FLIP ELEMENTS FLIP CURVES  
FLIP SURFACES TABLES  
ALL SELEC VISIB OUTL TOP  
EXIST UNSEL INVIS SURF BOT  
SELECT SET END LIST (\*)  
RETURN MAIN

MESHED BODY  
SHOW PROPERTIES STRUCTURAL  
APPROACH VELOCITY  
ANISOTROPIC FRICTION  
WEAR  
BOUNDARY DESCRIPTION DISCRETE  
OBSOLETE PROPERTIES  
RESET OK

MAIN  
CONTACT  
CONTACT BODIES  
NEW  
TYPE: MESH (DEFORMABLE)  
NAME  
Type e.g. small\_end\_cb  
PROPERTIES  
DISCRETE  
OK

# Contact

## Contact bodies: small\_end\_cb

The screenshot shows a CAD software interface with two main panels. The left panel is titled 'CONTACT BODIES' and has a dropdown menu set to 'STRUCTURAL'. It contains a 'NEW' button, a 'REM' button, and a text field for 'NAME' containing 'small\_end\_cb'. Below this is a 'TYPE' dropdown set to 'Meshed (Deformable)'. There are also buttons for 'COPY', 'PREV', 'NEXT', 'EDIT', and 'PICK'. The 'PROPERTIES' section shows 'MODEL SECTIONS' with 'ADD' and 'REM' buttons and a value of '0', and 'ELEMENTS' with 'ADD' and 'REM' buttons and a value of '192'. The 'CONTACT BODY VISIBILITY' section has a 'DETECT MESHED BODIES' button. At the bottom of this panel are buttons for 'ID CONTACT', 'ID BACKFACES', 'FLIP ELEMENTS', 'FLIP CURVES', 'FLIP SURFACES', 'TABLES', 'ALL:', 'SELEC.', 'VISIB.', 'OUTL', 'TOP', 'EXIST.', 'INVIS.', 'SURE', 'BOT', 'SELECT', 'SET', 'END LIST (#)', and 'RETURN'. The 'SET' button is highlighted with a yellow box. The right panel is titled 'CURRENTLY DEFINED SETS' and contains a table with the following data:

Set Name	Type	Count
bush	curve	4
bush_elems	element	128
conrod	curve	17
conrod_id	curve	135
conrod_id_surfaces	surface	27
gudgeon_pin	curve	4
gudgeon_pin_elems	element	801
pocket_elems	element	319
shank_elems	element	403
small_end_elems	element	192

At the bottom of the right panel is an 'OK' button highlighted with a yellow box. Below the panels is a 3D model of a mechanical part with a mesh. At the very bottom is a toolbar 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'.

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

The screenshot displays the 'CONTACT BODIES' panel in a CAD application. The 'ANALYSIS CLASS' is set to 'STRUCTURAL'. The 'NAME' is 'small\_end\_cb' and the 'TYPE' is 'Meshed (Deformable)'. The 'ELEMENTS' count is 192. The 'DETECT MESHED BODIES' section has 'ID CONTACT' checked. A red arrow points to the 'ID CONTACT' checkbox. Another red arrow points to the 'small\_end\_cb' color swatch in the legend. The main view shows a meshed part with a semi-circular cutout, where the inner boundary is highlighted in pink. The bottom toolbar includes options like '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'.

CONTACT BODIES  
ANALYSIS CLASS  
STRUCTURAL  
NEW REM  
NAME small\_end\_cb  
TYPE Meshed (Deformable)  
COPY FREE NEXT EDIT PICK  
PROPERTIES  
MODEL SECTIONS ADD REM 0  
ELEMENTS ADD REM 192  
CONTACT BODY VISIBILITY  
DETECT MESHED BODIES  
ID CONTACT ID BACKFACES  
FLIP ELEMENTS FLIP CURVES  
FLIP SURFACES TABLES  
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

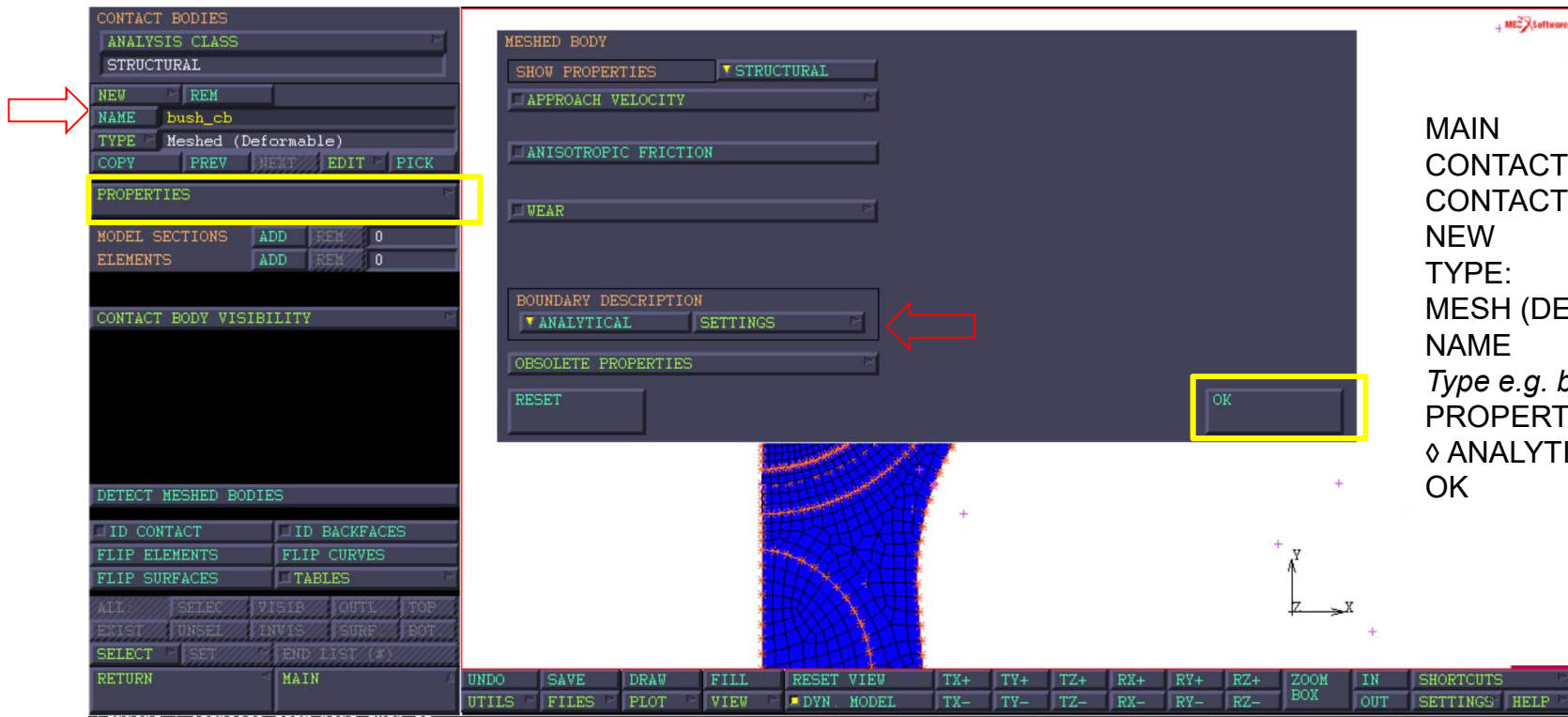
MAIN  
CONTACT  
CONTACT BODIES  
NEW  
TYPE: MESH (DEFORMABLE)  
NAME  
Type e.g. small\_end\_cb  
PROPERTIES  
DISCRETE  
OK  
ELEMENTS: ADD  
SET  
e.g. small\_end\_cb  
OK

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

The screenshot displays a software interface with several panels. On the left, the 'CONTACT BODIES' panel shows 'bush\_cb' with type 'Meshed (Deformable)'. The 'ELEMENTS' section has 'ADD' highlighted in yellow. The 'CONTACT BODY VISIBILITY' panel is below. The main 'MESHED BODY' panel shows 'ANALYTICAL' selected under 'BOUNDARY DESCRIPTION', with 'SETTINGS' highlighted in yellow and a red arrow pointing to it. The 'ELEMENTS' section of this panel has 'ADD' highlighted in yellow. The bottom of the interface shows a coordinate system and a toolbar with buttons like 'UNDO', 'SAVE', 'DRAW', 'FILL', 'RESET VIEW', 'TX+', 'TY+', 'TZ+', 'RX+', 'RY+', 'RZ+', 'ZOOM BOX', 'IN', 'OUT', 'UTILS', 'FILES', 'PLOT', 'VIEW', 'DYN. MODEL', 'TX-', 'TY-', 'TZ-', 'RX-', 'RY-', 'RZ-', 'ZOOM BOX', 'IN', 'OUT'. A red arrow points to the 'NEW' button in the 'CONTACT BODIES' panel.

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:  $\diamond$  MANUAL

2-D NODES: ADD

Select manually the bush nodes located at the symmetry plane ZY with normal direction X=0 labelled as:

3251

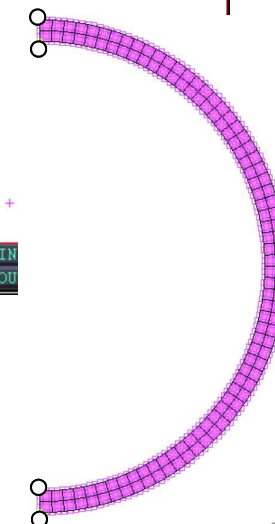
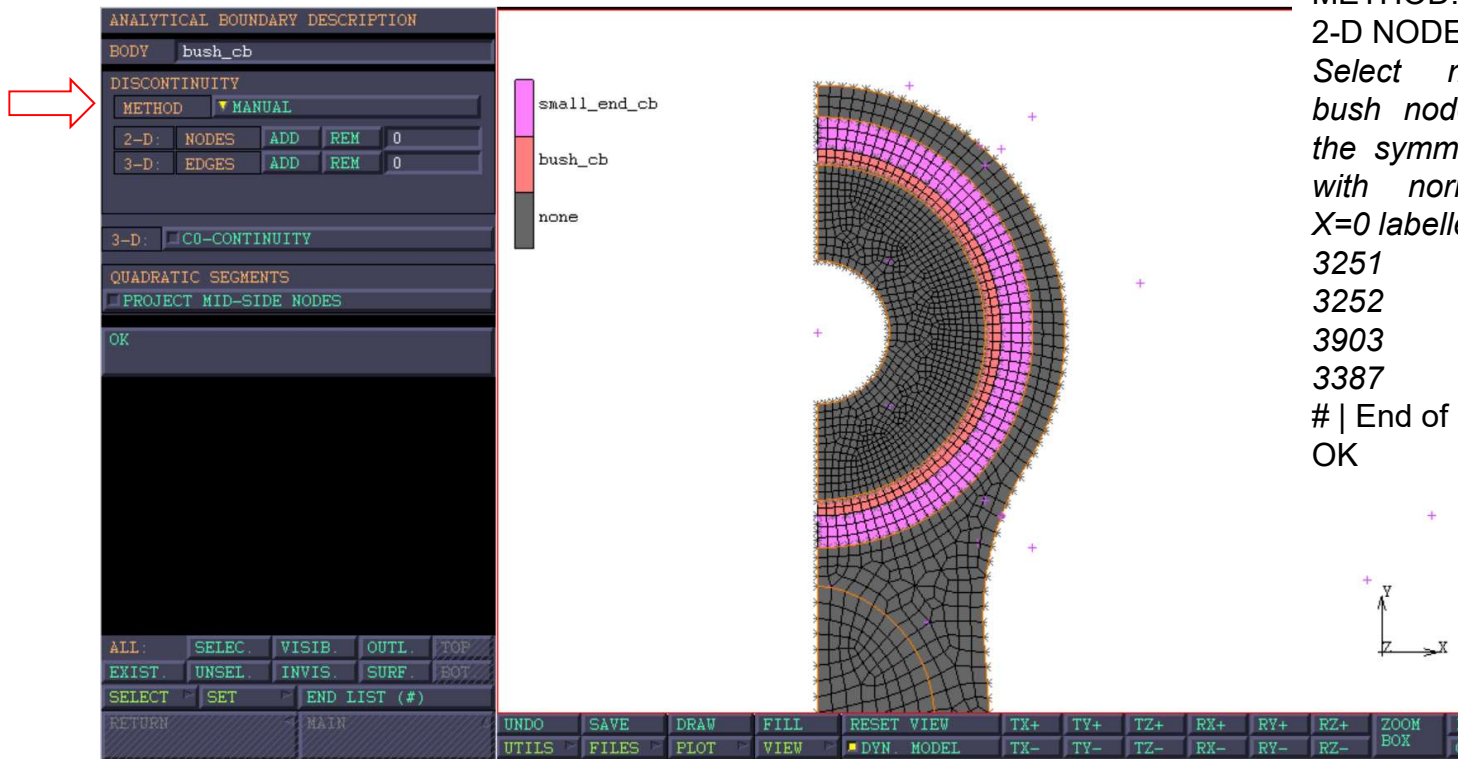
3252

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:

The screenshot shows the 'CONTACT INTERACTIONS' menu with the following details:

- CONTACT INTERACTIONS**
  - ANALYSIS CLASS: STRUCTURAL
  - NEW: REM
  - NAME: unilateral\_frictionless
  - TYPE: Meshed (Deformable)
  - COPY: PREV, NEXT, EDIT
  - PROPERTIES:
    - MERGE DUPLICATE INTERACTIONS
    - REMOVE UNUSED INTERACTIONS
    - REMOVE ALL INTERACTIONS
  - TABLES
  - ALL, SELEC, VISIB, OUTL, TOP
  - EXIST, UNSEL, INVIS, SURF, BOT
  - SELECT, SET, END LIST (\*)
  - RETURN, MAIN
- CONTACT INTERACTION TYPE** (highlighted in yellow)
  - MESHED (DEFORMABLE) vs. MESHED (DEFORMABLE)
  - MESHED (DEFORMABLE) vs. GEOMETRIC (WITH NODES)
  - MESHED (DEFORMABLE) vs. GEOMETRIC
  - MESHED (DEFORMABLE) vs. SYMMETRY
  - GEOMETRIC (WITH NODES) vs. GEOMETRIC (WITH NODES)
  - GEOMETRIC (WITH NODES) vs. GEOMETRIC
  - GEOMETRIC (WITH NODES) vs. SYMMETRY
  - CANCEL

The bottom toolbar includes: UNDO, SAVE, DRAW, FILL, RESET VIEW, TX+, TY+, TZ+, RX+, RY+, RZ+, ZOOM BOX, IN, SHORTCUTS, OUT, SETTINGS, HELP.

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:

The image shows two panels from the Marc Mentat software interface. The left panel, titled 'CONTACT INTERACTIONS', shows a list of interactions under the 'STRUCTURAL' analysis class. The 'unilateral\_frictionless' interaction is selected, and its properties are displayed in the right panel. The right panel, titled 'CONTACT INTERACTION PROPERTIES', shows the following settings:

- NAME: unilateral\_frictionless
- TYPE: Meshed (Deformable)
- CONTACT TYPE: TOUCHING (highlighted in yellow)
- CONTACT TOLERANCE: REDEFINED, VALUE: 0.001 (highlighted in yellow)
- BIAS FACTOR: DEFAULT
- CONTACT TYPE: TOUCHING (highlighted in yellow)
- CONTACT TOLERANCE: 0.001 (highlighted in yellow)
- OK button (highlighted in yellow)

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

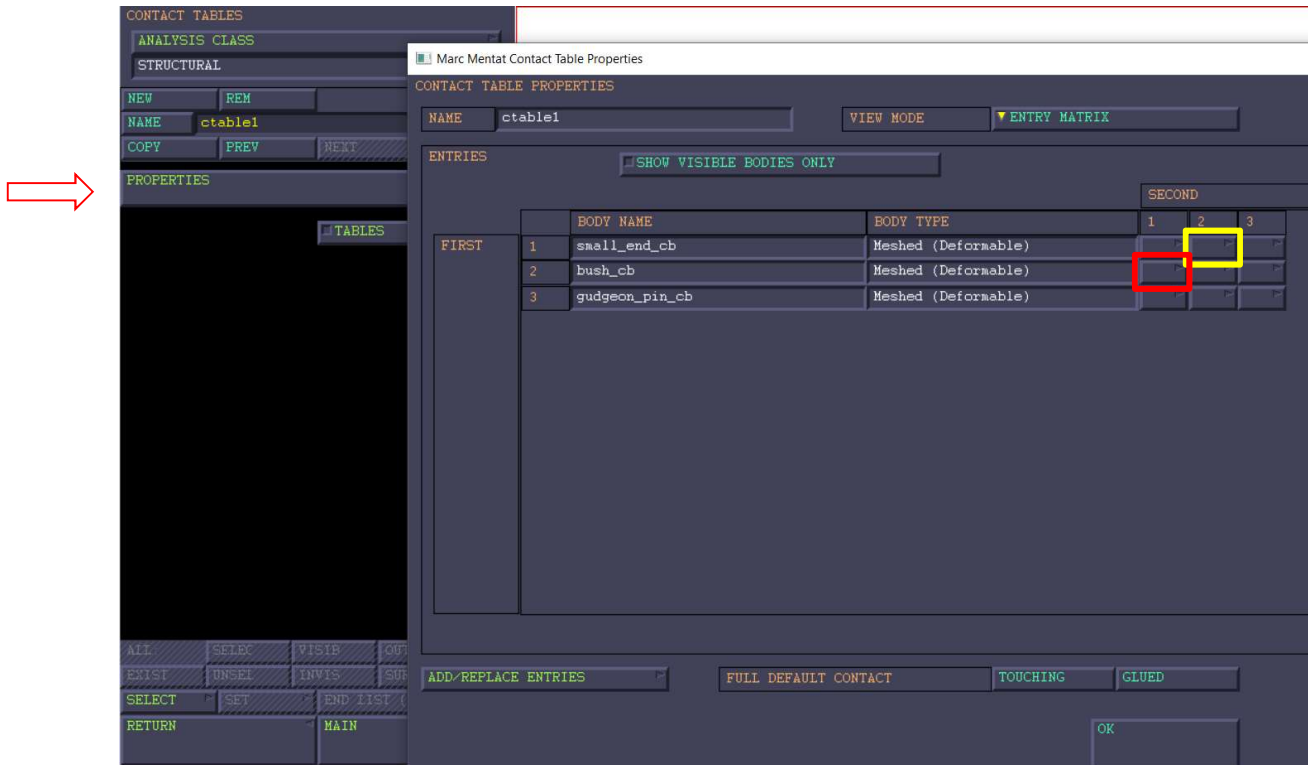
To define the contact between the bodies under scrutiny:

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

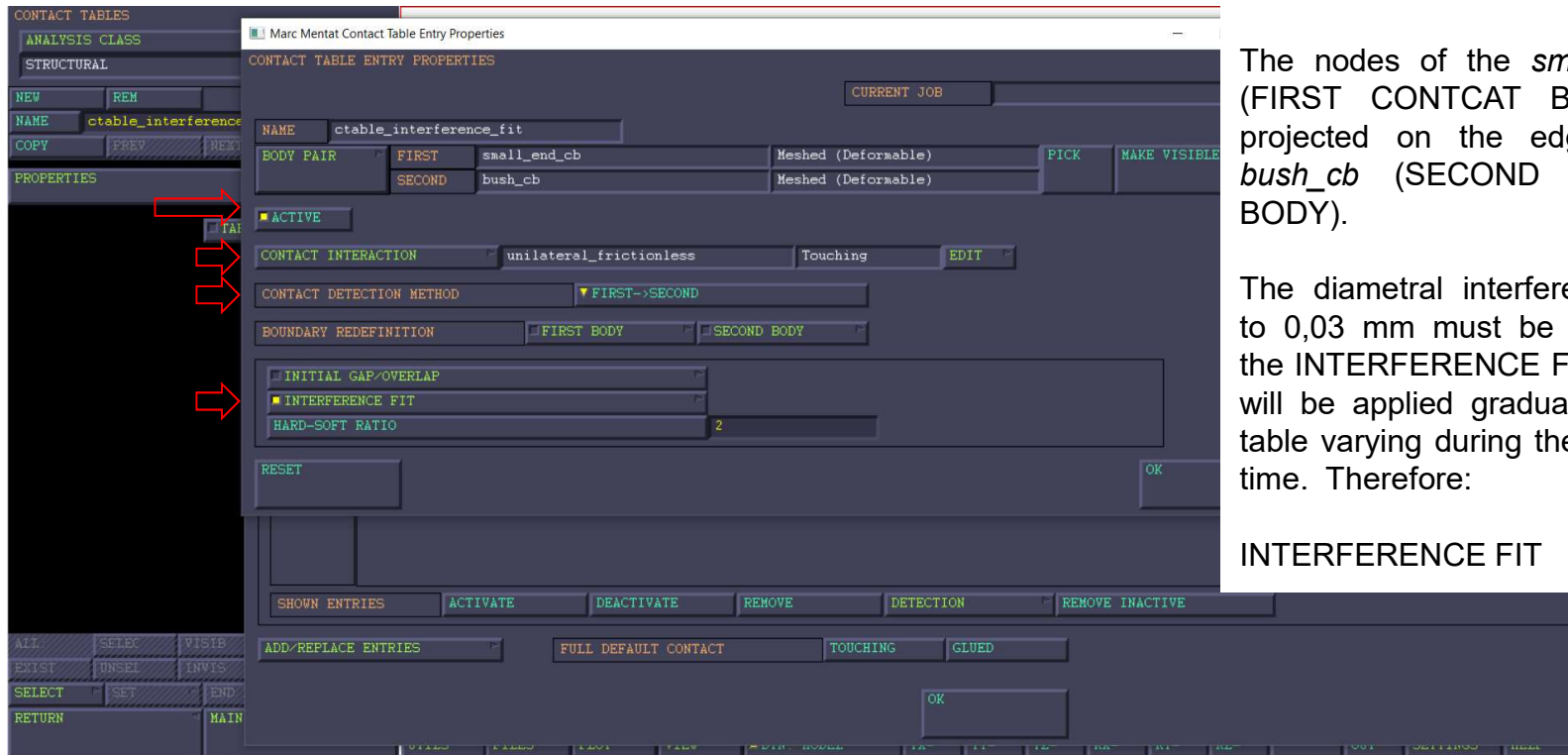
CONTACT INTERACTION  
Upload the *unilateral\_frictionless*  
contact interaction defined  
previously

CONTACT DETECTION  
METHOD:  
FIRST → SECOND

The nodes of the *small\_end\_cb*  
(FIRST 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



# Contact

## Contact table

CONTACT INTERACTION  
Upload the *unilateral\_frictionless*  
contact interaction defined  
previously

CONTACT DETECTION  
METHOD:  
FIRST → SECOND

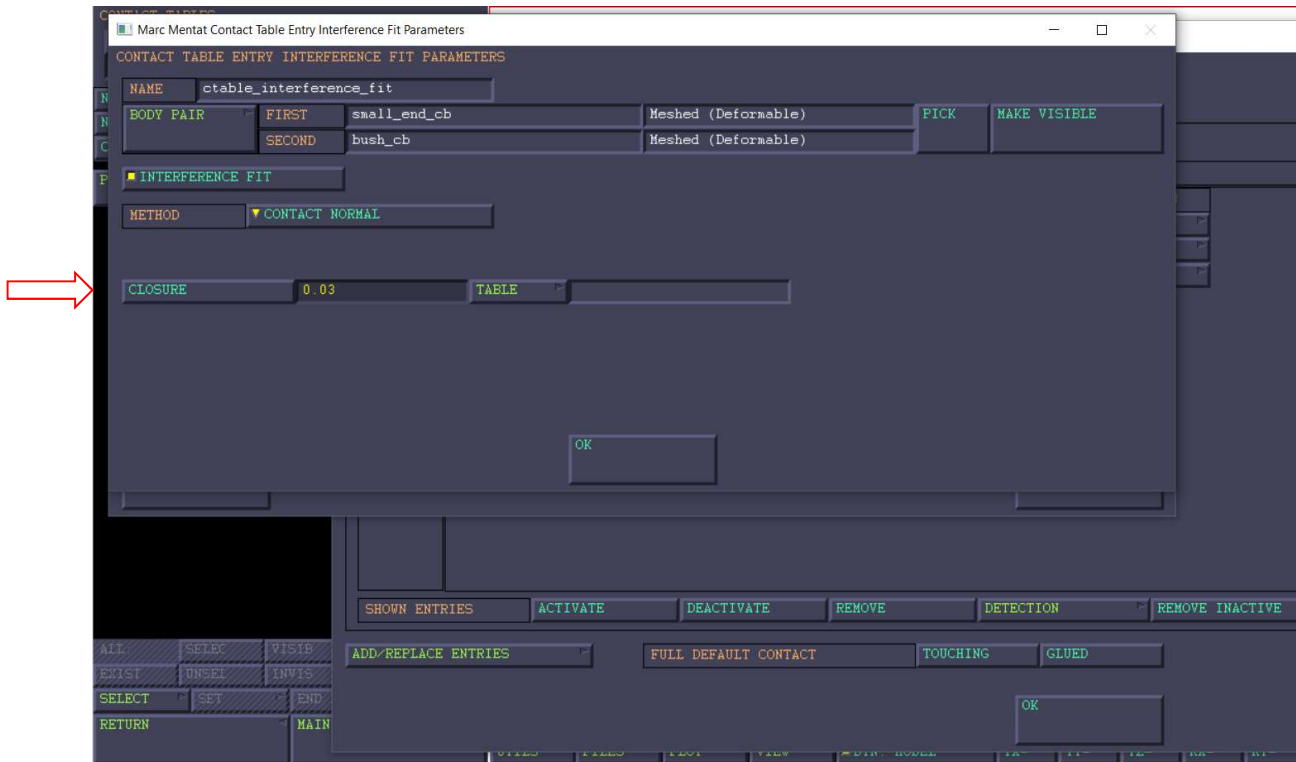
The nodes of the *small\_end\_cb*  
(FIRST CONTACT BODY) are  
projected on the edge of the  
*bush\_cb* (SECOND CONTACT  
BODY).

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

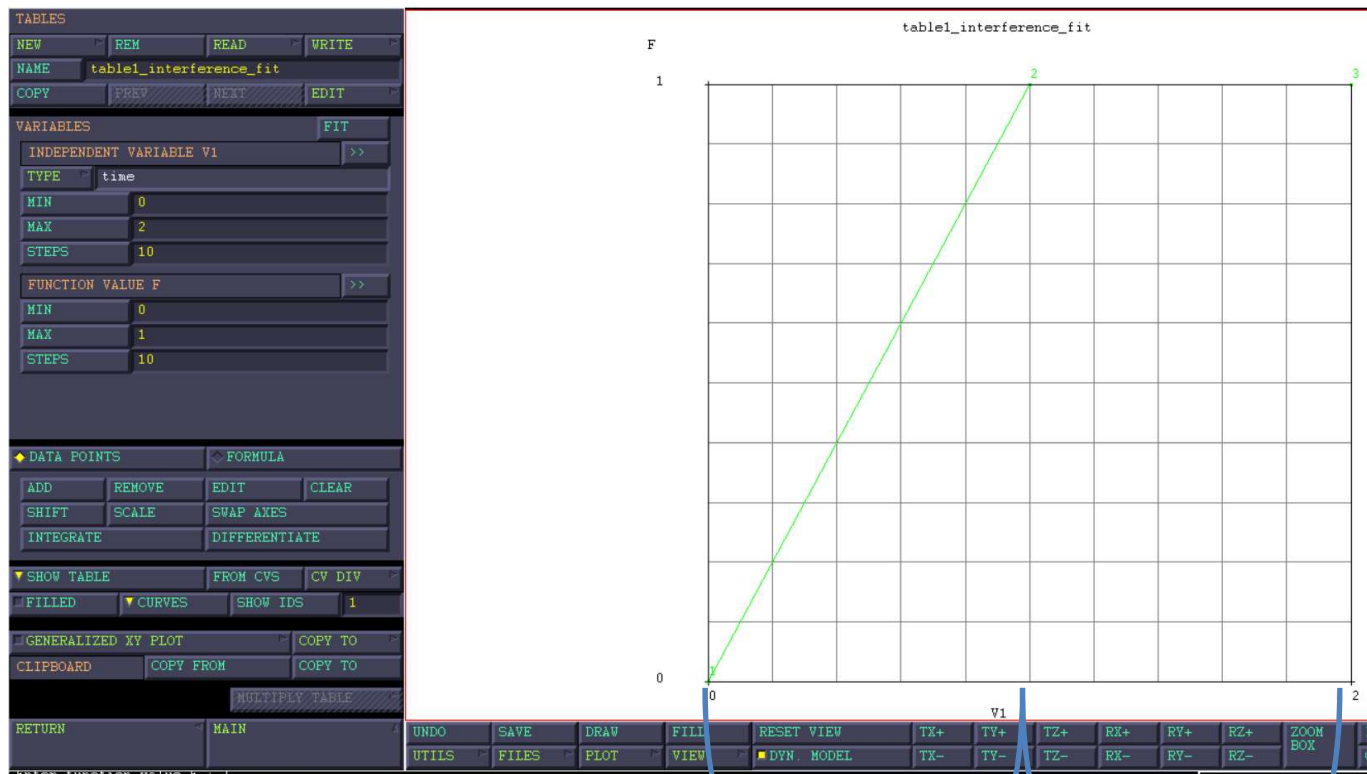
## Contact table

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  
INDEPENDENT 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 piecewise 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.

1 sec

2 sec

# 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:  $z = f(v1, v2, v3, v4)$

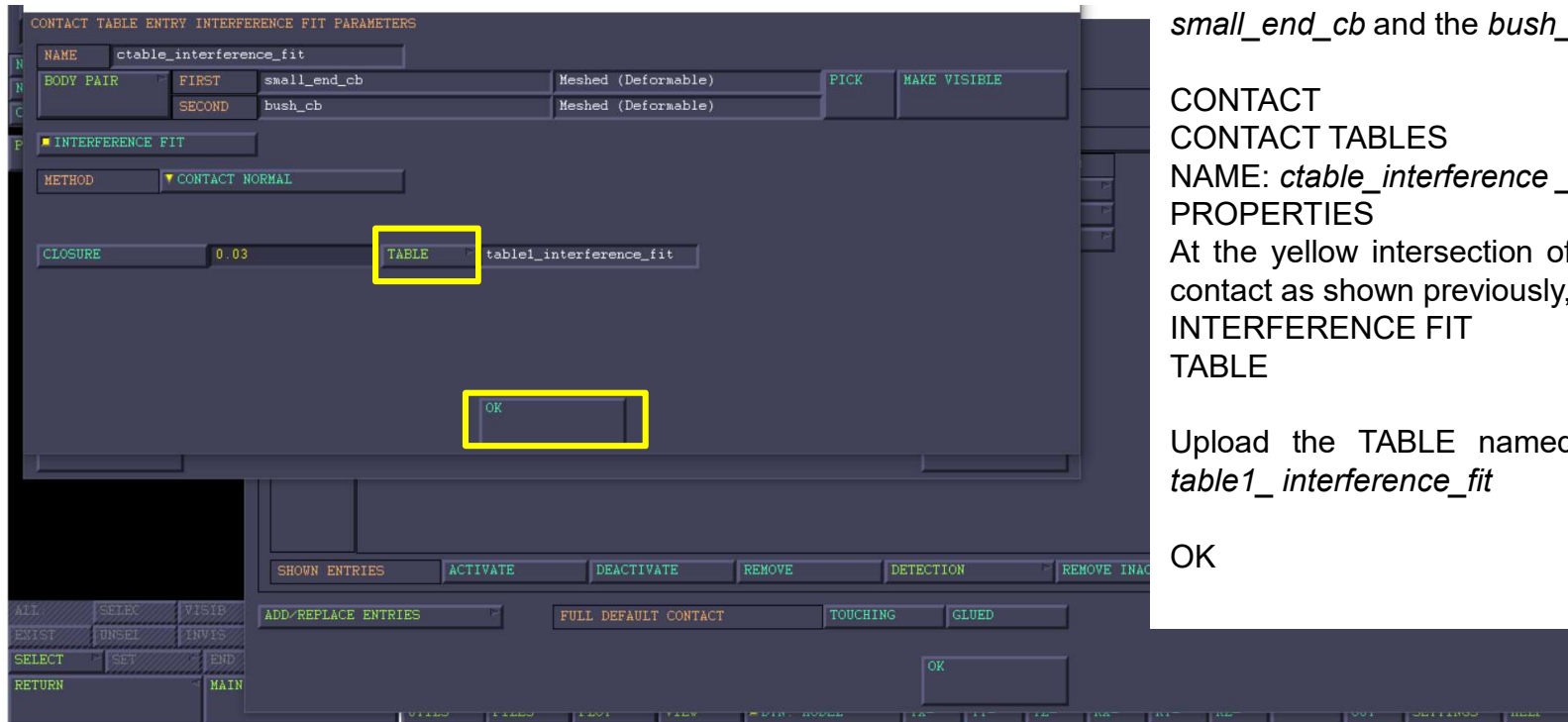
nindep = 1, ndep = 2:  $z = f(v1), z2 = f2(v1)$

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

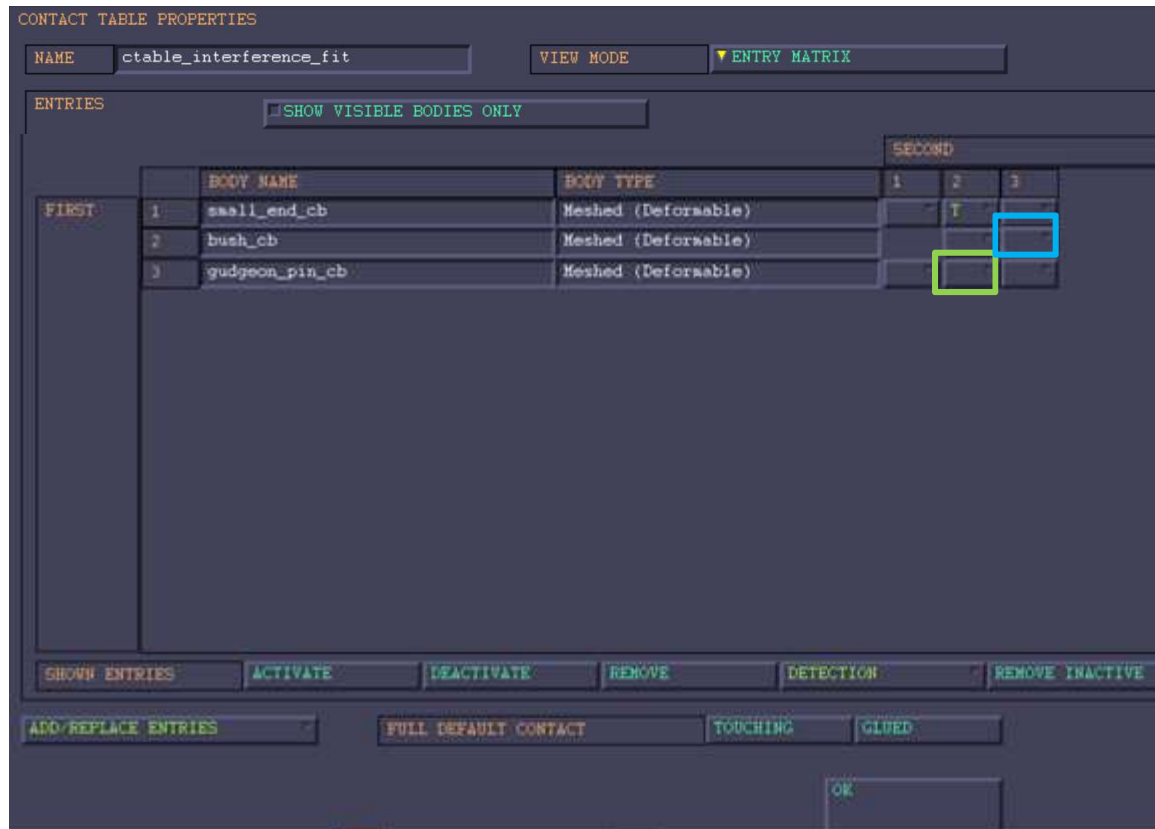
At the yellow intersection of the  
contact as shown previously, T  
INTERFERENCE FIT  
TABLE

Upload the TABLE named as  
*table1\_interference\_fit*

OK

# Contact

## Contact table



Now, we define the CONTACT between the *bush\_cb* and the *gudgeon\_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 TABLE ENTRY PROPERTIES

CURRENT JOB

NAME *ctable\_interference\_fit*

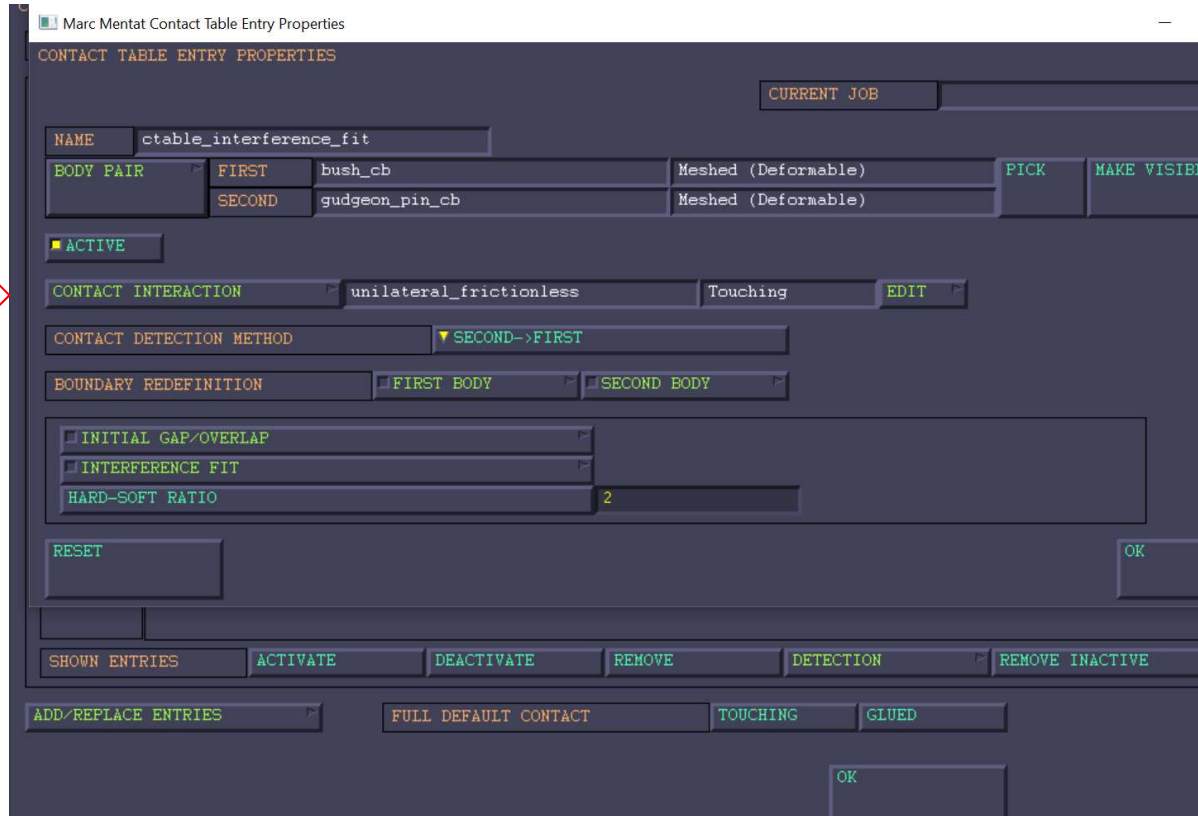
BODY PAIR	FIRST	<i>bush_cb</i>	Meshed (Deformable)	PICK	MAKE VISIBLE
	SECOND	<i>gudgeon_pin_cb</i>	Meshed (Deformable)		

ACTIVE

RESET OK

# Contact

## Contact Table



### CONTACT INTERACTION

Upload the *unilateral\_frictionless* contact interaction previously defined.

CONTACT DETECTION METHOD:  $\diamond$  SECOND  $\rightarrow$  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 clearance between the bush and the pin will be considered in the following.

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

Now Deflag ACTIVE!!!!

# Contact

## Contact table

### CONTACT INTERACTION

Upload the *unilateral\_frictionless* contact interaction previously defined.

CONTACT DETECTION  
METHOD:  $\diamond$  SECOND  $\rightarrow$  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 clearance between the bush and the pin itself will be considered in the following.

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

Now Deflag ACTIVE!!!!

OK



# Contact table

## Contact Table: Summary

CONTACT TABLE PROPERTIES

NAME:  VIEW MODE:

ENTRIES  SHOW VISIBLE BODIES ONLY

FIRST		BODY NAME	BODY TYPE	SECOND		
				1	2	3
FIRST	1	small_end_cb	Meshed (Deformable)		T	
	2	bush_cb	Meshed (Deformable)			
	3	gudgeon_pin_cb	Meshed (Deformable)		-	

SHOWN ENTRIES    DETECTION

# 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):

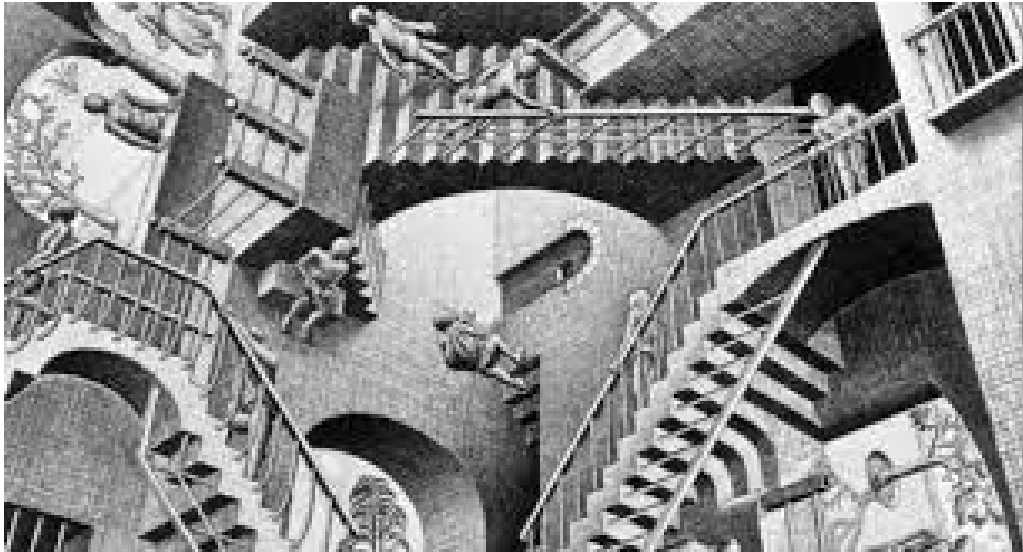
- Plane stress and plane strain, pp. 138-157;
- Pressure vessels, pp. 657-679;
- Shaft-hub press-fit, pp. 690-700;
- Contact problems, pp. 501-518.

## **Maxima:**

- pf\_lame.wmx → analytical evaluation 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.