

**A Guideline for PDMS
Electronic Spec.
Generation**



شرکت مهندسی پتروپالامحور

R&D Department

**A Guideline
For**

PDMS

**Electronic Spec.
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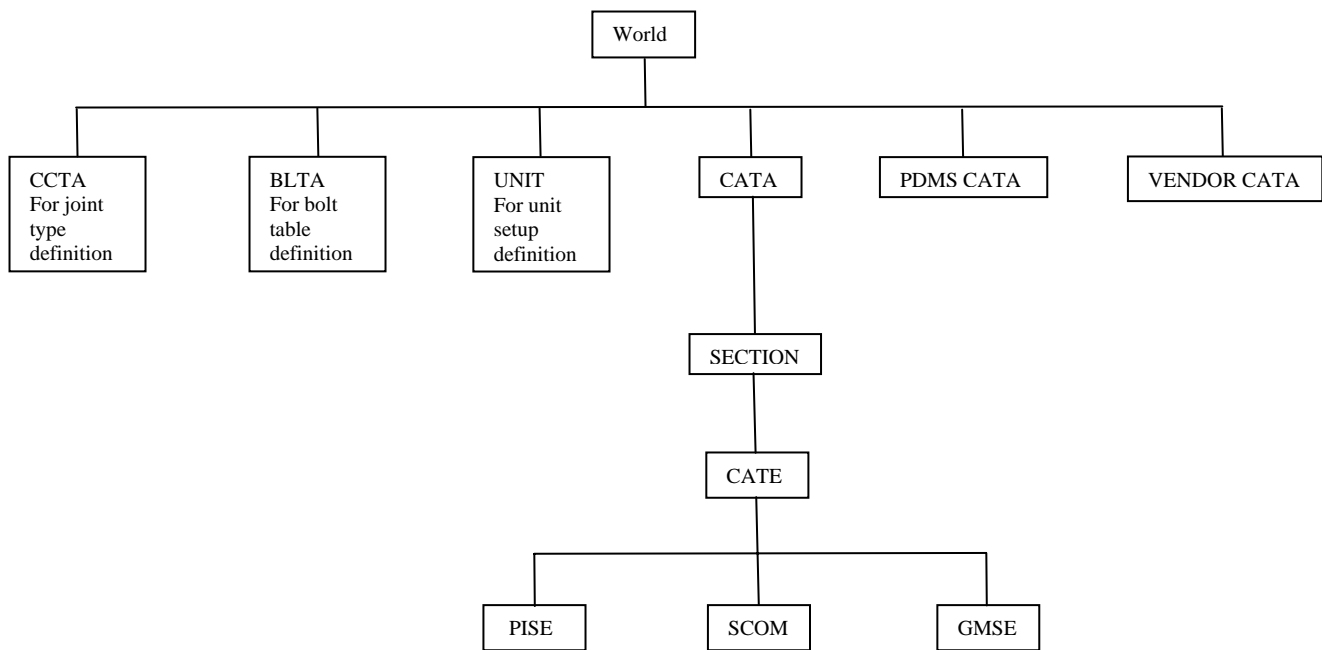
Prepared by:

F a r s h a d S a r a e i

1. PDMS modules that is used for electronic spec. generation

- PARAGON module for elements geometrical and joint type definition
- SPECON module for generated elements inserting in the project electronic spec.

2. PDMS catalogue hierarchy



3. Definition of connection types

- Connection type normally shall be defined in three characters as : ☒ - 1 - 2 - 3
- 1 for “Type”, for example : F/flange type , G/gasket , W/wafer type
- 2 for “Facing”, for example : A/flat face , B/raised face
- 3 for “Rating”, for example : A/125# , B/150# , C/300#
- ☒ Only used for wafer type elements

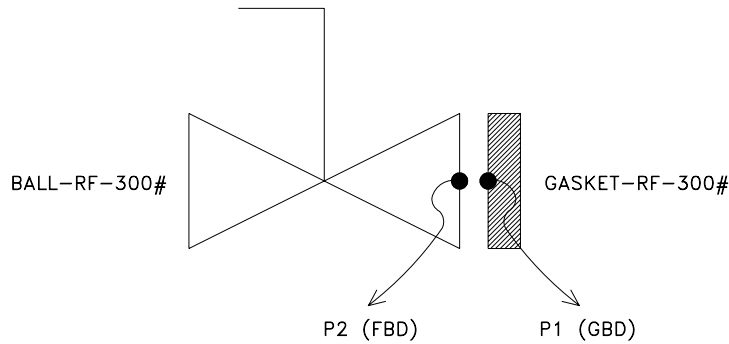


Figure-01

4. COCO table

- This table clears, which type of connections can be jointed together. For example as it seems in Figure-01, “FBD” can be jointed with “GBD”.
- It should be avoided to define a new connection type in COCO table, out of the PDMS standard connections, because the software calculates the required number and length of bolts due to it. For example when the software receives to a “WFBD” connection, automatically understands, that is a wafer type connection and the length of the attached valve shall be provided in the bolt length calculation.

5. Example-1 , Creating an Elbow (4”-90°)

- Project : ILC
User : Admin
Password : 1234
MDB : Offsite
Module : Paragon
- 1) Create : CATA (ILC-CATA)
- 2) Create : SECT (ILC-ELBOW)
- 3) Create : CATE (ELBOWSA)

At this step it shall be defined “P.Point” , “Geom. Set” , “PTSE name” and “GMSE name” in the hierarchy.

- 4) Create : /ELBOS.PT
- 5) Create : /ELBOS.GM

A Guideline for PDMS Electronic Spec. Generation



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At this step it shall be defined the component main parameters. For an elbow it can be defined as below :

PA1 = nominal bore
PA2 = connection type
PA3 = leg length
PA4 = outside diameter
PA5 = weld sim. DIA (diameter of sphere in center line)

Now, we can make the related component as below :

- 6) Create → Component : /ELBOW4 (for 4")
- 7) Selection of "Geometric Type" (This is so important for the reports preparation)

Note 1 : For welded joints, the connection type is "BWD"

Note 2 : The above mentioned activity (no.6), shall be "n" times repeated for "n" sizes.

Note 3 : The above command (no.6) makes different "SCOM"s.

- 8) Display → Component (With this command, it can be appeared each "SCOM" that we are set on it in the related hierarchy.

For making point sets in hierarchy, it shall be followed as below :

- 9) Create → Point set → Primitives → Axial P.Point

At this window, we should start to insert the P.Points information. For example:

- PARA1 : Bore
- PARA2 : Connection type
- Etc.

Note 4 : It's better, the "Display" window will be kept open during make of "SCOM"s, because at this window we can automatically shift to PTSE and GMSE in hierarchy with going on PTREF or GMREF.

For making point sets in hierarchy, it shall be followed as below :

- 10) Create → Geometry set → Primitives

At this window, it shall be created three Geometries for each element as below :

- A) Piping detail
- B) Center line
- C) Obstruction

Note 5 : Some elements such as elbows, don't require to any obstruction space, but some another items such as valves, have require to provide obstruction space for them.

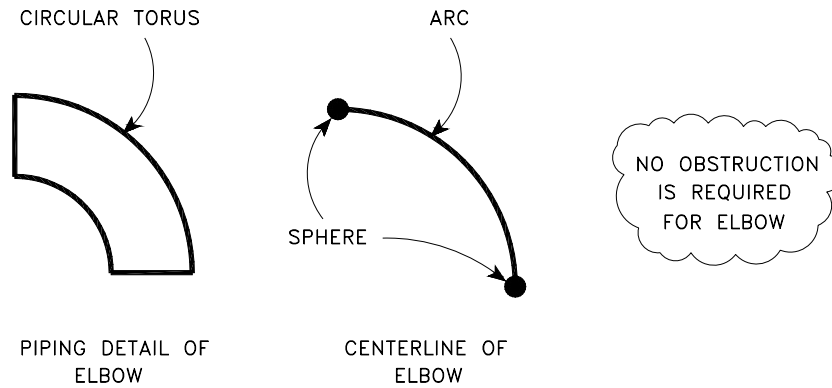


Figure-02

It can be used of “Circular Torus” for the elbow 3d-shape. Also the center line can be made of one “Arc” that is mixed by two “Spheres”.

Please pay attention, it’s better to define X , Y , Z based on the P.Points. for example :

- X axis : P1
- Y axis : P2
- OD : PARA4

Note 6 : When we make the “Center line” for one size (SCOM), It will be made the required “Center line” for the another sizes (SCOMs) automatically after it’s “Piping detail” generation.

6. Example-2 , Required parameters for an ECC Reducer generation

PA1 = large nominal bore
PA2 = small nominal bore
PA3 = connection type
PA4 = large OD
PA5 = small OD
PA6 = length
PA7 = weld sim. DIA (diameter of sphere in center line)

Note 7 : It is possible to use of mathematical symbols and numbers to make some complex parameters. For example :

- (PA2 + PA5)
- (2 * PA7)
- (- PA2)
- Etc.

These complex parameters shall be started and ended with “Parentheses” and it shall be used “Spaces” between each mathematical symbol and the name of simple parameters.

7. Example-3 , Creating a Flange (4”-RF-300#)

The required parameters shall be defined as below :

- PA1 = nominal bore
- PA2 = P1 connection type
- PA3 = P2 connection type
- PA4 = OD of flange
- PA5 = thickness of flange
- PA6 = OD of pipe
- PA7 = OD of hub
- PA8 = length of hub
- PA9 = weld sim. DIA (diameter of sphere in center line)

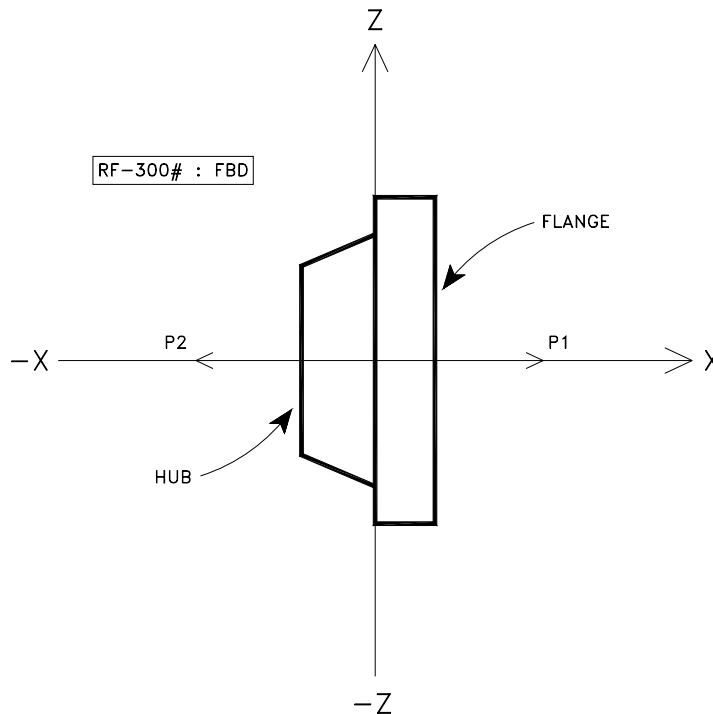


Figure-03

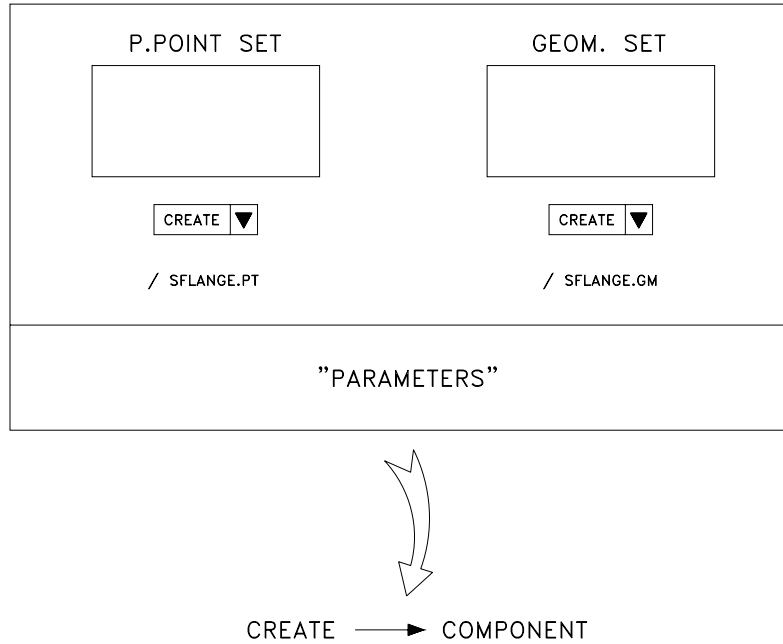


Figure-04

In the above window (Figure-04), flange shall be selected as "Generic type". Then it shall be selected the "PTSE" and "GMSE" names, for example in case of 4" flange, we can select the names as "SFLANGE4" and then we can enter the required parameters based on piping TC.

Note 8 : It's better that "PSKEY" will be set as "NULL" .

To generate of flange center line model, we can follow the below schematic :

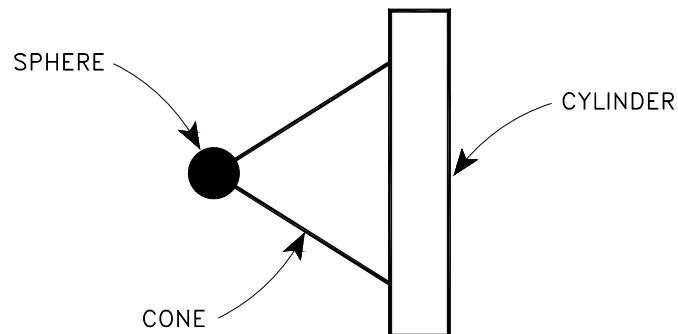


Figure-05

Also to generate of flange obstruction model (due to maintenance requirements of bolt and nut removal spaces), we can use a box as below :

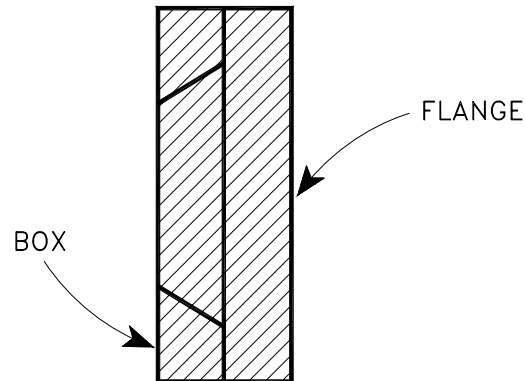


Figure-06

Note 9 : If we want to set the PDMS 3d-model obstruction mode as “off “ in design module and only see a special item (such as a flange or valve or ...) obstruction, we shall reduce it’s “Drawing levels” in the Geom.set window during generation of that element in the catalogue module (for example from 6~8 to 0~6).

Note 10 : To avoid of PDMS software hang, it shall be generated a unique “P.point set” and “Geom. Set” for each CATE. It means, we should define a separate CATE for each group of elements (such as Flange, 90°Elbow, 45°Elbow,...).

Note 11 : In each CATE, we have a unique “P.point set” and “Geom.set” that all the related “SCOM”s will set themselves by it. Then we can only generate one size of element and for the other sizes only change the parameters based on the piping TC. The software will generate all the required models of the other sizes (detail, center line and obstruction) automatically.

8. Example-4 , Creating a Spiral wound gasket (4"-RF-300#)

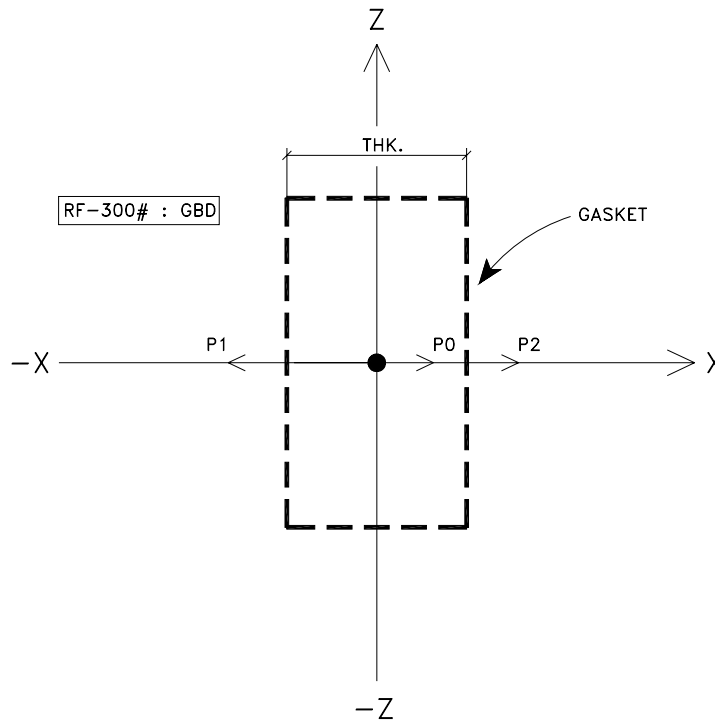


Figure-07

The required parameters shall be defined as below :

PA1 = nominal bore
PA2 = connection type
PA3 = thickness

Note 12 : It's not required to define any "Geom.set" for gaskets (because based on the PDMS standard it shall be provided a hollow space instead of gaskets). Then the creation of gasket will be completed by "P.point set" definition only.

Note 13 : The connection name of any wafer type elements that is connected to a flange, shall be started with "W" (except gaskets) :

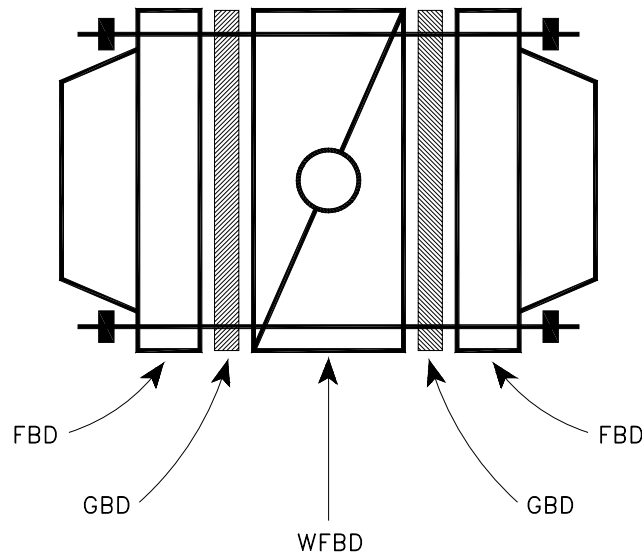


Figure-08

9. Example-5 , Creating a Pipe

The required parameters shall be defined as below :

PA1 = nominal bore

PA2 = connection type

Connection type : TUBI

GTYPE (generic type) : TUBE

10. Example-6 , Creating an Attachment

Attachment is used for making a note, support, steam tracing or etc. in PDMS 3d-model.

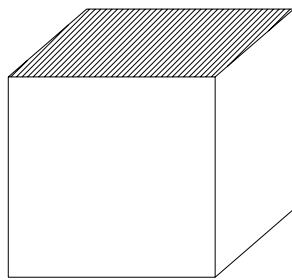
The required parameters shall be defined as below :

PA1 = nominal bore

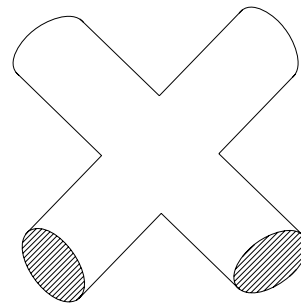
PA2 = connection type

Connection type : TUBI
GTYPE (generic type) : Attachment
GMSE : Box or Sphere or ... can be used

Note 14 : The GMSE of an attachment can be defined as below schematics :



BOX



TWO CYLINDERS

Figure-09

11. Example-7 , Creating a Weld

The required parameters shall be defined as below :

PA1 = nominal bore
PA2 = connection type

Connection type : TUBI
GTYPE (generic type) : Weld
GMSE : Sphere or a thin Cylinder can be used

Note 15 : The selected GTYPE in paragon module shall be exactly same as the selected GTYPE in SPEC. (Specon module).

Note 16 : For quick generation of an element, we can going to PDMSPIPE.CATA in the members window and then going into the required category (for example Gate valve) and select one of it's related "SCOM"s. then we can use of this command : Modify → Component to see the required parameters and create the new model based on that.

12. Example-8 , Creating a Weldolet or Socolet

The required parameters shall be defined as below :

- PA1 = main pipe nominal bore
- PA2 = branch connection type
- PA3 = main connection type
- PA4 = run connection
- PA5 = main outside diameter
- PA6 = run outside diameter
- PA7 = olet height
- PA8 = weld sim. DIA (diameter of sphere in center line)

Connection type P1,P2 : BWD or TUBI

Connection type P3 : BWD or SWF or SCF

Generic type : Tee or Olet

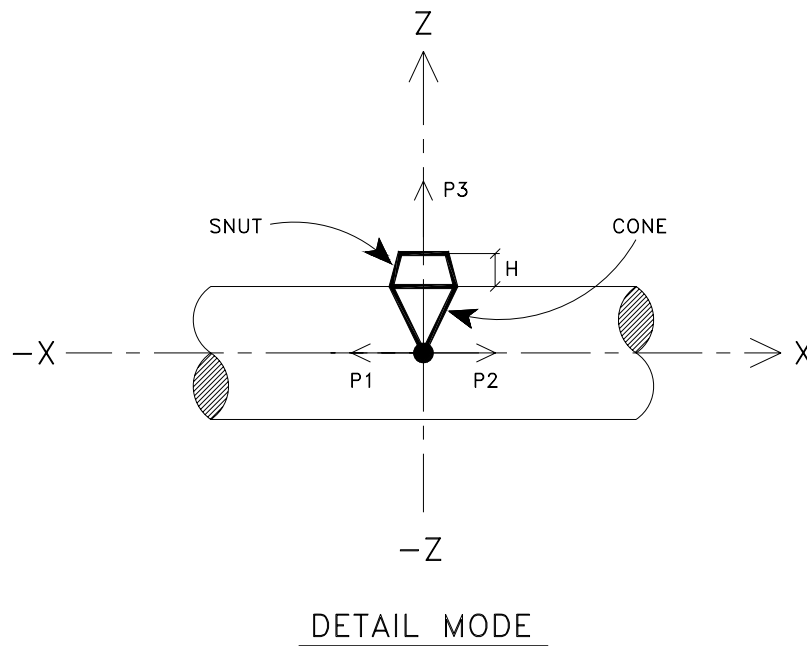
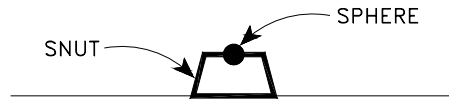


Figure-10



CENTER LINE MODE

Figure-11

Note 17 : In screwed and socket welded items, the “Geom. set” shall be adjusted to be greater than “P.point set” dimensions.

Note 18 : Three types of obstruction is available :

- 0 : means no obstruction
- 1 : means soft obstruction
- 2 : means hard obstruction

It's better that “2” will be selected as default for all generated elements because we want to be informed about any clashes in our 3D-models.

13. Example-9 , Creating a pipe to pipe connection

The required parameters shall be defined as below :

- PA1 = main nominal bore
- PA2 = branch nominal bore
- PA3 = main connection type
- PA4 = run connection
- PA5 = main outside diameter
- PA6 = run outside diameter

Connection type P1,P2 : BWD
Connection type P3 : BWD
Generic type : Tee

Note 19 : The “DDANG” parameter, is defined in paragon module, But it can be modified in design module (change of angle) with this command too : Modify → Attribute

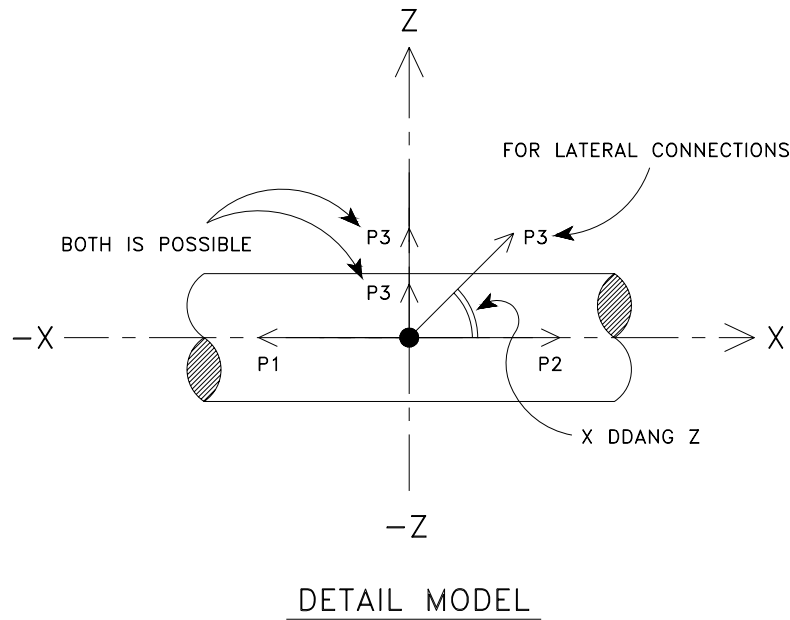


Figure-12

Note 20 : Setting \longrightarrow Model parameters

It's possible to have a visual test from the generated elements in paragon module by using of the above mentioned command. After running this command, we can enter the related parameters (such as bore, angle, etc.) and see the generated elements without going to design module.

The entered numerical parameters will not be saved due to this command. (it is a visual test only).

Note 21 : Normally in pipe to pipe connections, the "P3" shall be adjusted on 90° (+Z). if any designer wants to model a pipe to pipe connection with the another angle (such as $30^\circ, 45^\circ, 60^\circ$), he/she should change the angle by using of this command : Modify \longrightarrow Attribute

14. Example-10 , Creating a fixed radius and variable angle elbow

The required parameters shall be defined as below :

Generic type : Elbow

P1,P2 : BWD

Circular torus : from P1 to P2

P1 : PDIR = -Y

PDIS = TANF R DDANG

P2 : PDIR = +Z
PDIS = TANF R DDANG

Note 22 : It shall be defined a detail text (DETREF) for any generated elements in paragon module. This detail text will be inserted in the bill of material part of ISOMETRIC drawing.

To define of detail text, we should going in to “DETREF” window. This window will give to us three choices as below :

- A) RTEXT
- B) STEXT
- C) TTEXT

PDMS software automatically use of the RTEXT as a default to preparing of ISOMETRIC drawings, but it is possible to replace it with STEXT or TTEXT in “ISO ADMIN” definition.

We shall make the “DETREF” of each element under it’s related “CATE” same as this example :

CATE —> 90° Elbow
Create —> Detail Text

CREATE	▼	/90° Elbow-D	
	▼		▼
		Butt Welded Elbow	
Elbow			
	▼		▼
Symbol Key <input type="text" value="Elbow"/>			
Length <input type="text"/> Quantity <input type="text"/>			
RTEXT	ELBOW 90DEG ANSI B16.9 BW		
STEXT	For additional notes (if will be required)		
TTEXT	For additional notes (if will be required)		

Figure-13

Note 23 : If it will be inserted a number in “Quantity” field, the software automatically multiples the quantity of the related item to this number. This may be useful in some cases such as spare part quantity calculation.

14. Material definition

Each element in design module, has a specific material that is identified by a parameter as “Material Text” (Attribute : MATXT).

XTEX : ASTM A105 (default)

YTEX : (optional)

ZTEX : (optional)

During isometric drawings generation, “XTEX” is considered as elements material by default.

Note 24 : Type of material shall be input in “Material Text” only. Unless otherwise (if it will be input in “DETREF”), the material report of elements can not be generated correctly.

For material definition, we shall make a “SECT” under the related “CATA” in hierarchy, that will be our total library of materials. For example :

World → ILC-CATA → ILC-MATERIAL

Command : Create → Material Standard

With running of the above mentioned command, we can input infinite number of materials in “XTEX” field.

Note 25 : The PDMS software has no sense regarding to “SCHEDULE”. Then it shall be defined in detail text.

15. The procedure of COCO table generation

World

Command line : NEW CCTA / ILC-COCO

Command line : NEW COCO / ABD-DCA

Command line : Q ATT

Command line : : CTYPE → ABD-DCA

With the above mentioned comments, it is generated a COCO table, that’s name is “ILC-COCO” and we can define any nonstandard connection on it (such as ABD to DCA).

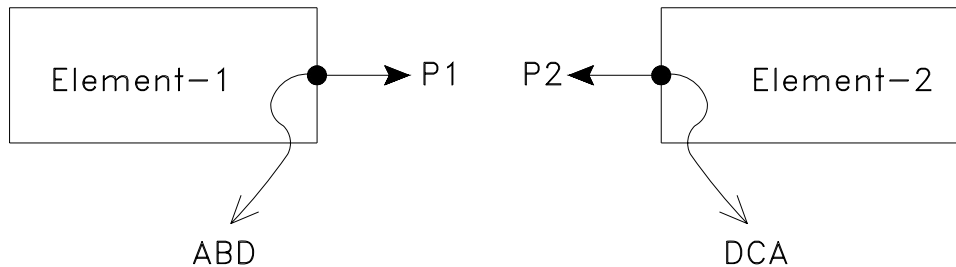


Figure-14

Note 26 : Utility \longrightarrow Modify Ckey

With using of this command and push the “CE” button, we can see the total list of PDMS standard connections.

Note 27 : If two elements can not be connected together in design module, it may be caused by these problems :

- Their connection types are not match together due to COCO table in paragon module.
- Their bores are not match together due to specon module.

16. Some of important parameters

Paragon Module	Design Module
DDANG	Angle
DDHEIGHT	Height
DDRAD	Radius
DESPARA	Design Parameters

For example if “DESPARA” is used in the structure of element, in paragon module, we can replace it with the numerical information and make the element in design module based on our required dimensions. For example, if this parameter used in valve hand wheel, we can change it’s dimension in design module due to size of valve.

“n” numbers of “DESPARA” can be used in the structure of elements. Default of these parameters is zero and can be replaced with numerical information during detail 3d-modeling.

For example :

(PARA1 + PARA5 + DESPARA1)

(PARA7 – DESPARA1 + DESPARA2)

17. Pipe supports generation

The pipe support can be made in paragon module as an attachment, which parameters are defined with some DESPARA's and it's geometry is set as box or cylinder or etc.

In design module, we can give any numerical content to DESPARA's by using of this command :

Modify → Attribute , and create the 3D shape of supports. Also the Tag-no. of support can be defined in "DETREF" field of attachment.

In this method, due to use of attachments instead of supports, we will not have any report of clash between pipe and supporting system and the real shape of supports can be shown in 3D-model too.

18. PDMS SPECON module

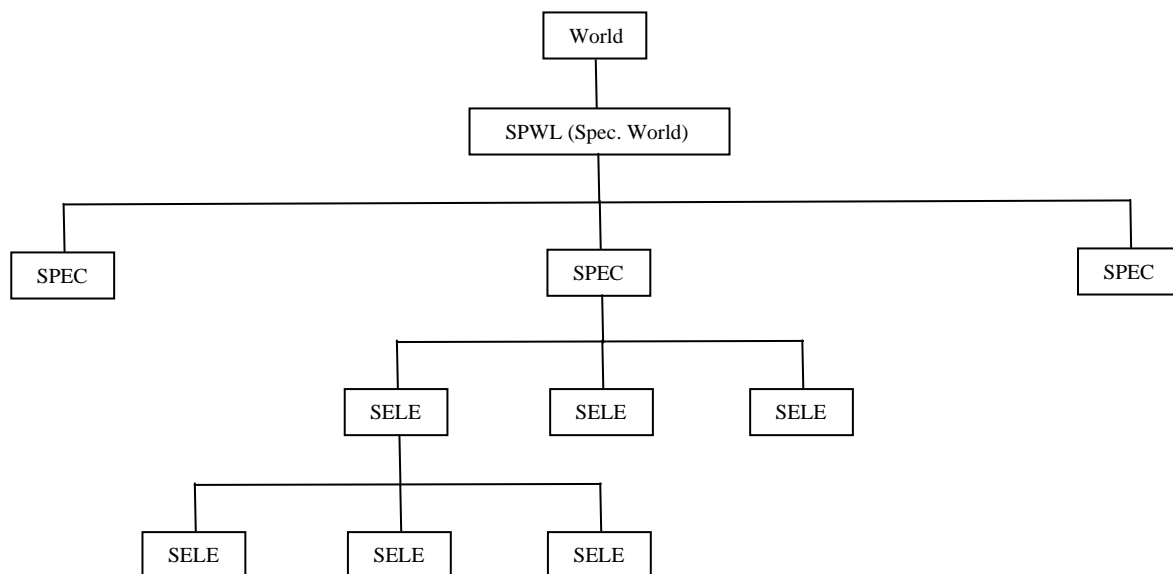
This module is used for electronic spec. generation in PDMS software. The name of electronic spec. shall be exactly same as it's related name of class, in the PMS of "Iran LNG" project.

CATA → SECTION → CATEGORY → SCOM1 , SCOM2 , ...

- ✓ CATREF = name of SCOM
- ✓ DETREF = detail text name
- ✓ MATXT = material text name
- ✓ BLTREF = bolt reference name
- ✓ STYP = component design name
- ✓ SHOP = erection or fabrication

19. PDMS SPECON hierarchy

The hierarchy of SPECON module is same as below schematic:



Note 28 : The number of “SELE” is depended to number of Bores.

At the first, it shall be made a “SPWL”. For example :
*/World → SPWL → ILC-SPEC

20. Example-11 , Spec. generation for a Flange (Class no.A1A2V)

We shall going to “Word Pad” and write as below :

20.1. New specification /A1A2V (Enter)

20.2. Text ‘Piping’ (Enter)

20.3. Heading (Enter)

20.4. Type name pbor Φ shop styp catref detail matxt cmpref bltref (Enter)

20.5. Defaults (Enter)

20.6. - - 0.00 = = (Enter)

20.7. Flan */Flan4 100 true text ‘Flange WN-RF 300#’ /Sflang /Flan-D A105 =0 =0 (Enter)

20.8. \$\$+ (Enter)

20.9. Then we should save the above mentioned MACRO (with the name as Flange) and going out from the “Word Pad” window.

20.10. Now, we should return to Specon module : Paragon → Specon , and run the above MACRO :
*\$M/C:\Flange (Enter)

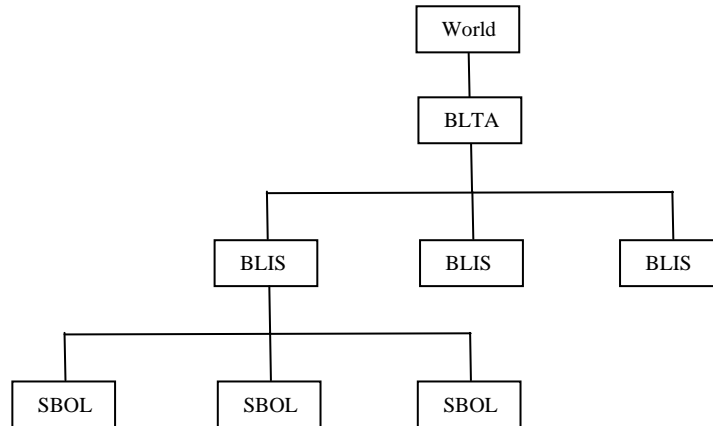
Note 29 : To run any MACRO file in PDMS software space, it shall be written : \$M/Path (Enter)

Note 30 : If it will be required to going out from Specon module due to any problem, we can write in command line :
* Quit Specon

20.11. * Savework (Enter)

20.12. Now, we shall going to Design module and check that, is the new generated Spec. coming into the design module or not ?

21. Hierarchy for Bolt & Nut Spec. generation



An example of bolt and nut information definition is same as below :

Bolt information :

BDIA : 12mm
Length : 70mm
NOFF : 4
NSTD : /M-12

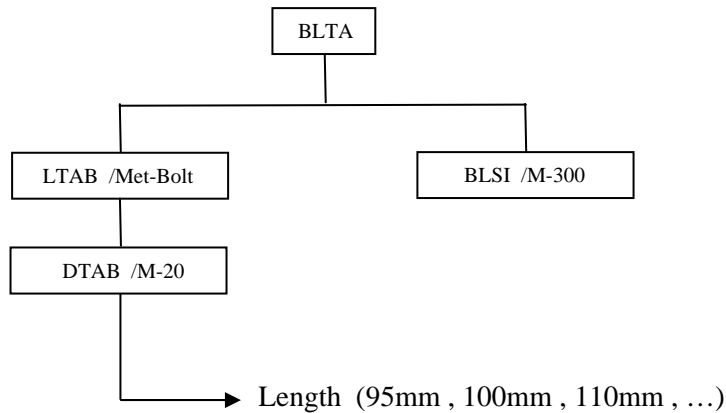
Nut information :

BITEM	Nut	Wash	Wash	Nut
BITLENGTH	20	2	2	20

22. Example-12 , Spec. generation for Bolt & Nut (Machine Bolt 4"-RF-300#)

BDIA : 20mm
Length : 100mm
NOFF : 8
NSTD : /M-20
BITEM : Wash Wash Nut
BITLENGTH : 2 2 20

It shall be generated a Bolt length table (BLTA), which is included all the allowable length of the bolts in M-20 category :



WORLD

BLTA : ILC-BOLT
BLIS : M-300-Mach
SBOL : MSSSB
LTAB : MET-Bolt-L
DTAB : M-20

At this step we shall get a “Q ATT” and set the standard and nonstandard bolt length parameters as below :

- * Boltlength (95mm 100mm 110mm)
- * Nstdblength /M-20

Note 31 : All the elements generation is done in “Command Line” window, for example :

New BLTA (Enter)
New LTAB (Enter)
Etc.

For input of numerical parameters based on the piping TC, we shall run “Q ATT” command after making each element and change it’s related parameters simply.

Note 32 :

For stud Bolts : Nut Wash Wash Nut
For machine Bolts : Wash Wash Nut

Then we shall going to SPECON module :

- Specification A1A2V
- * Ffile / C: \ Flange
- * Out Flange / A1A2V Term

At this time it shall be opened the Flange created file in the “Windows Text Editor” space and put (/ MSSSB) on it. then we can save and exit and return to Specon module to run the above MACRO file as below :

- * \$M / C: \ Flange (Enter)
- Savework (Enter)
- * Paragon (Enter) or * Design (Enter)

Note 33 : In Design module we can going on the created Flange and write: Q BLTREF of SPREF
By this action, the software refers the BLTREF to MSSSB. It means, the bolt is putting on the created flange.

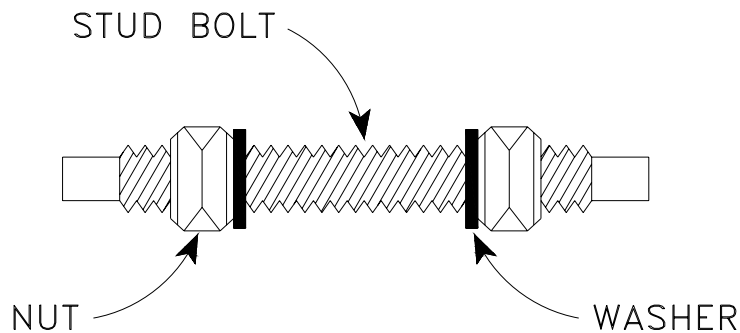


Figure-15

23. Insulation

In each insulated element, a parameter as “IPARA” shall be added due to it’s insulation. This parameter can be variable in Design module, by any change in “Bore” and “Temperature”. The obstruction of insulation is automatically set as “1” (soft clash).

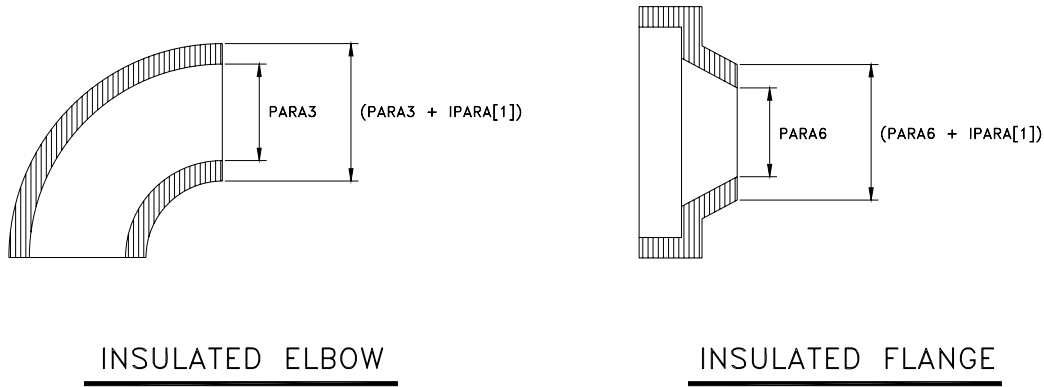


Figure-16

We shall enter to CATA space (ILC-CATA) and create a new section as : ILC-INSU and then create it's SCOM as below :

- New Scom / I-25 Gtype Insu Param 50
- New Scom / I-40 Gtype Insu Param 80
- New Scom / I-50 Gtype Insu Param 100
- New Scom / I-60 Gtype Insu Param 120

Now, it shall be created a new text file in C: as INSU.txt (we can get a copy from the previous file of Flange and modify it for insulation) :

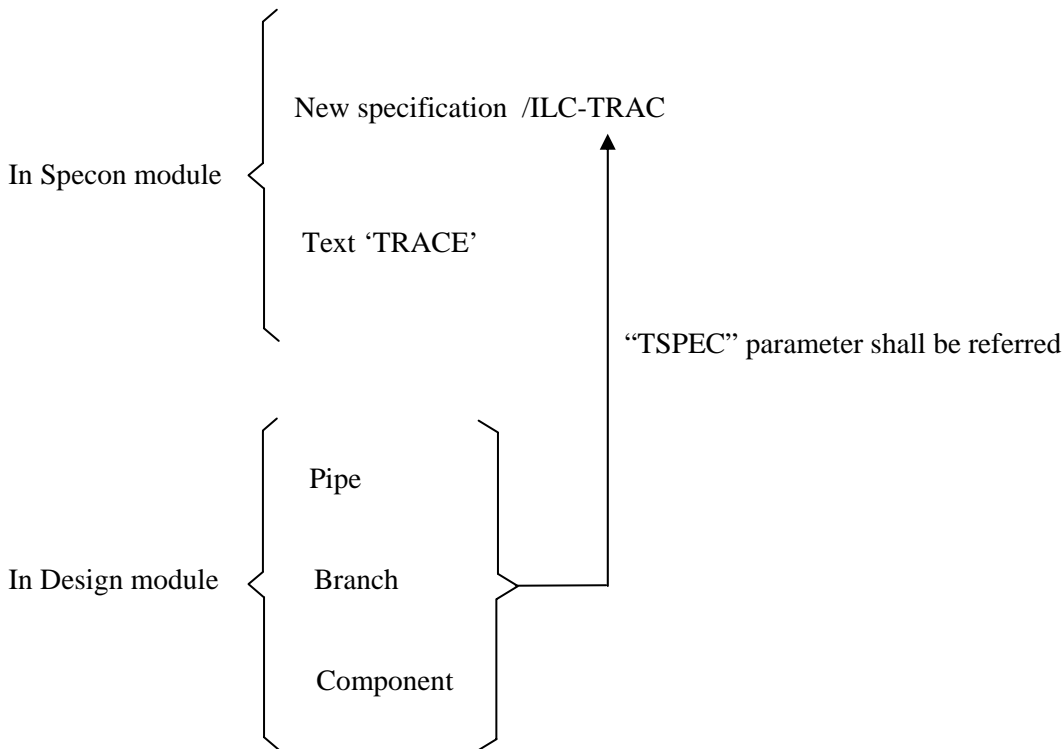
- New specification / ILC-INSUL
- Text 'INSU'
- Heading
- Type Name Pbor0 Temp Catref
- Insu */I25 40,150 120,200 /I-25
- Insu */I50 40,150 201,400 /I-50
- Insu */I40 200,300 120,200 /I-40
- Insu */I60 200,300 201,400 /I-60
- \$\$+

Then, we should save and exit and going to Specon module to run the above MACRO file :

- * \$M / C: \ INSU.txt (Enter)
- Savework (Enter)
- Design (Enter)

Now, in Design module we can go to Pipe and Branch and refer its ISPEC parameters to ILC-INSUL by using of “Q ATT” command. Also the temperature of Pipe and Branch shall be set by “Q ATT” command too. At the end of this process, the related insulation will be shown in 3D-model.

24. Tracing



25. CATVIEW module

“CATVIEW” is a utility module of PDMS software that is used for quick spec. generation.

In Paragon space : Utility → Catview

To create an spec. by catview, at the first it shall be defined the name of spec. in catview window. This name shall be same as its related class in the project PMS :

Create → Specification (The name of spec.)

It can be find so many standard elements in the “TYPE” section of catview window. To see all standards (for example DIN), we should stand on CATA-DIN and get it’s name parameter by “Q ATT” and copy it, into the (Modify → Defaults) part of the main window of catview. Then we shall push the “Load” button. After this process, if we sign out from catview and sign in again, DIN can be appeared in the “Standard” window.

To add any item from the selected standard to spec., it’s enough to standing on that item and shift it to spec. by pushing on the “ADD” button (with our required sizes and materials).

In “Editing Specification-Spec” window, we can see all the added items. The parameters of the added items can be modified by us via selecting them one by one or pushing the “ALL” button too. (for example : Stype , Name , Dtext , ...)

Some times catview has mistake in the bolt dimension settings. To solve this problem manually, we shall going to the related item (for example : 4”-Flange-RF-300#) and edit it. To do this, we shall stay on SBOL in the member window, by pushing on “Go to bolt” button in the bolt section. Now it is possible to change the bolt parameters (such as length , number , ...) by using of “Q ATT” command based on the standard piping TC.

Note 34 : After any batch of elements adding from a related standard to our spec., Catview automatically provide “A” character instead of their Stype. We can select all items and change the Stype from “A” to our required sentence (for example : Con. Red. BW ANSI B16.9).

Note 35 : A large library of “Control Valve” catalogues, is accessible via the “INST” part of the catview module.

26. The method of Insulation splitting

To splitting the insulation along a part of the pipe, it shall be created two attachments and insert them into the each end of splitting area limits. Then we should going to “Member” window and stay on “ATTA” and change it’s “ATTYP” parameter to “INSU” by using of “Q ATT” command.

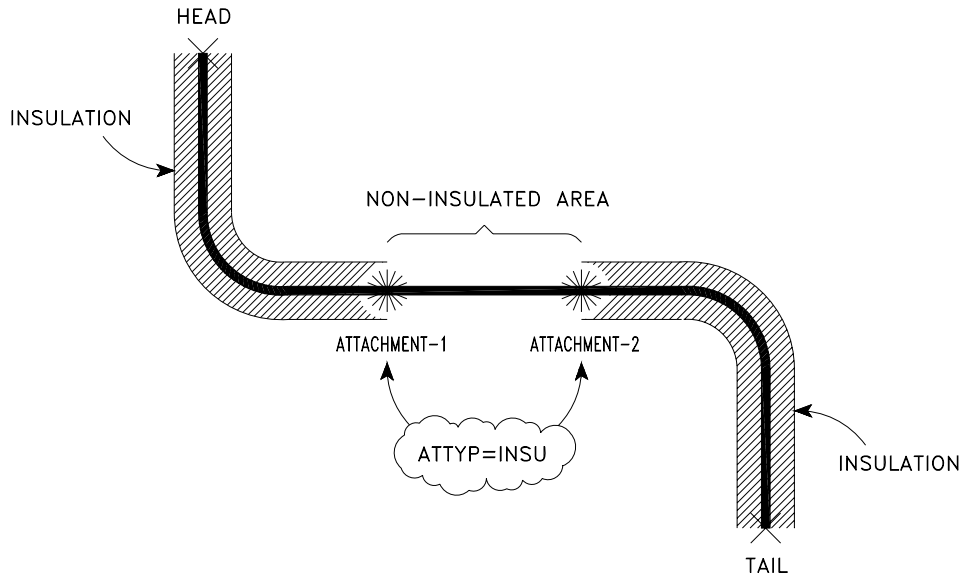


Figure-17

27. The method of Tracer splitting

The same procedure of “Insulation Splitting” can be followed with one change : ATTYP = TRAC

28. Writing a comment in 3D-model

It should be inserted an attachment in every location that we want to write a “Comment” in 3D-model. The “Stext” parameter of this attachment can be modified to our requested comment sentence by using of “Q ATT” command. This type of comments will be shown in Isometric drawings. For example :

Command line : Q ATT

Command line : Stext 'SAFE LOCATION'