Modelling Tube-to-Tube contact in Abaqus using Part and Instance

Key Words:
*Abaqus, ITT21 or ITT31, *Interface, *Slide Line, Tube-to-Tube, Tube in Tube contact*

When using Abaqus CAE, and especially Native mesh (imported mesh), you normally get Parts and Instances in your abaqus input file. Tube-to-Tube modeling is not supported in the Abaqus CAE pre-processor, so you will have to modify your inputfile manually. This can be done in a couple of steps.

Start out creating your model with beam, pipe or truss elements. Normally you would use Pipe Sections with predefined pipe properties, but in this example I am using Beam General Section.

Figure 1 show the model which consists of the inner pipe (L=2m) with OD 20in (508 mm) and wt = 0.835in (21.2 mm), and the outer pipe (L=1m) with OD 30in (762mm) and wt= 1.0in (25.4mm).

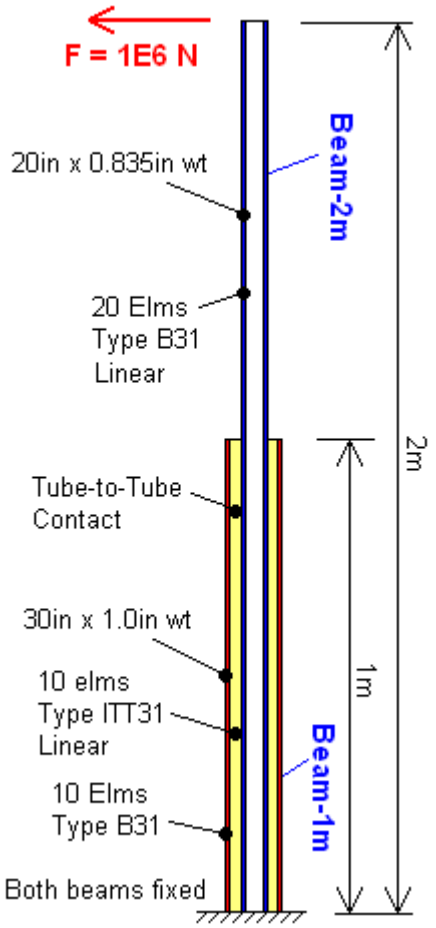

Figure 1. Tube-to-Tube model

In our model, the 2 m inner pipe is defined as the Slide Line, and the 1m outer pipe has got contact elements attached to its nodes. Only the 10 lower nodes of Beam-2m are included in the slide line, meaning that the upper part of Beam-2m is not in contact with Beam-1m. It doesn't matter if the mesh is native (imported into CAE) or modeled in CAE. You can see in the input file that Beam-1m has a native mesh.

Editing the inputfile

The Tube-to-Tube contact can be defined in 4 steps.

Step 1: Defining tube-to-tube contact elements

In the Assembly section of the input file, define the contact elements

```
*Element, type=ITT31, ELSET=TTT
11,  1          (Element number 11, Node number 1)
12,  2          (Element number 12, Node number 2)
13,  3
```

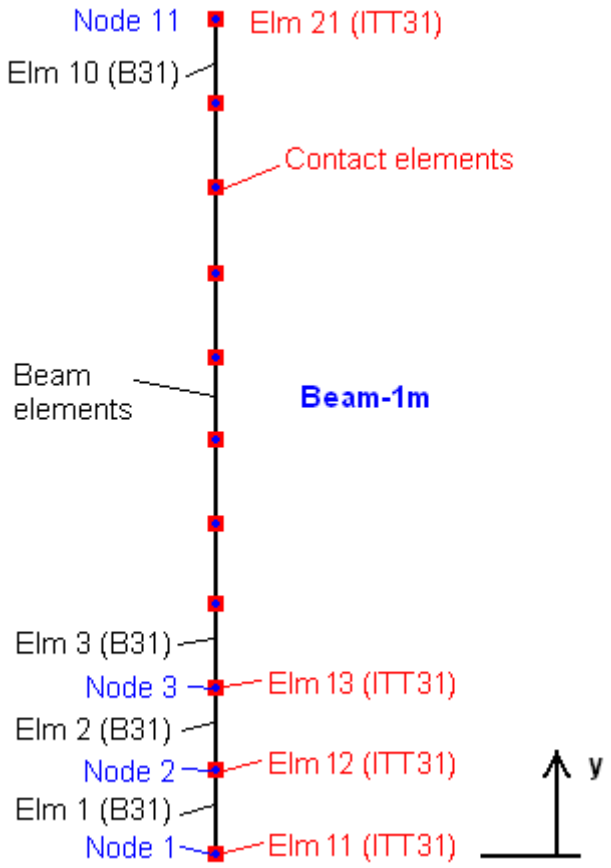

Figure 2. Defining contact elements ITT31

Use contact Element of Type=ITT21 together with 2-dimensional beam, pipe or truss elements, and Type=ITT31 together with three-dimensional beam, pipe or truss elements.

Elset = TTT will not be recognized outside the Instance. Remember to define later, see step 2.

Step 2: Defining element set for contact elements (optional)

Since the ELSET for the contact elements (ELSET=TTT) is defined inside the Instance, an ELSET will have to be created outside the Instance too, so that the element set can be recognized when connected to the slide line. In our model, we are using the same name (TTT).

```
*Elset, elset=TTT, instance=Beam-1m, generate
 11,21, 1
```

Step 3: Defining Element Properties for contact elements

Defining Contact element properties is very simple, since there is only the radial clearance that needs to be defined.

```
*Interface, Elset=TTT
0.001
```

Here, you are giving Elset=TTT defined above (the contact elements attached to Beam-1m) the radial clearance 0.001 m.

Normally, the radial clearance is equal to the space between the inner diameter of the outer pipe and the outer diameter of inner pipe, but in this case we have a liner or something in between. See Figure 3 for details on radial clearance.

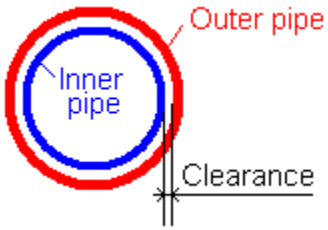

Figure 3. Radial Clearance

Step 4: Defining Slide Line

Finally, the slide line needs to be defined. The slide line consists of nodes only. In our case, the inner pipe is the slide line. Figure 4 show the Slide Line.

NOTE: Since the nodes of the Beam-2m is defined inside the part, we need to include the part name with the node numbers. This would not be necessary if you created the input file without parts and instances.

Choosing which pipe will have the slide line: In the case of internal tube-to-tube contact, the slide line can be placed on the inner tube or the outer tube. Generally the slide line should be associated with the outer tube. However, if the inner tube is stiffer than the outer tube, the slide line should be attached to the inner tube.
Note: In our model it was defined the "wrong" way, i.e. inner/ least stiff tube is slide line.

*SLIDE LINE, ELSET=TTT, TYPE=LINEAR
Beam-2m.1, Beam-2m.2, Beam-2m.3, Beam-2m.4, Beam-2m.5, Beam-2m.6, Beam-2m.7, Beam-2m.8,
Beam-2m.9, Beam-2m.10, Beam-2m.11

In the *Slide Line command, you are connecting the nodes in the slide line with ELSET=TTT.

Set TYPE=LINEAR to define a slide line made up of linear elements (B31), and TYPE=PARABOLIC to define a slide line made up of parabolic elements (B32)

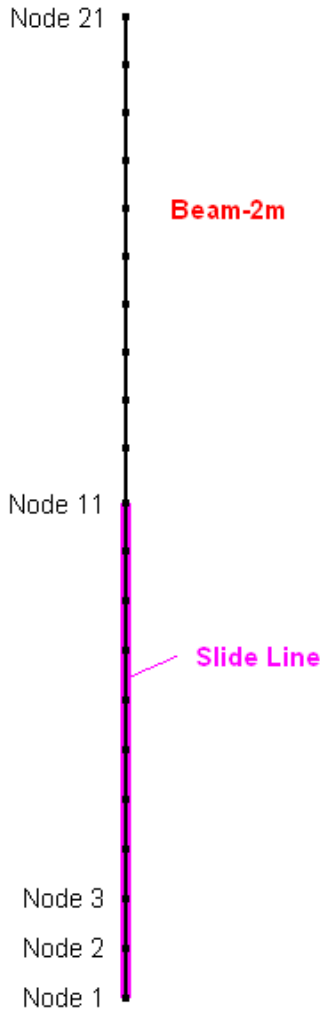

Figure 4. Slide Line

Results

Displacement on top of Beam-2m is listed in Table 1. Three radial clearance values have been run: 0.1 mm, 1 mm and 5 mm. Figure 5 show the deformation plot for 1 mm radial clearance.

Table 1. Deformation on top of Beam-2m

Radial Clearance	0.1 mm	1 mm	5 mm
Displacement on top	5.42 mm	6.41 mm	8.42 mm

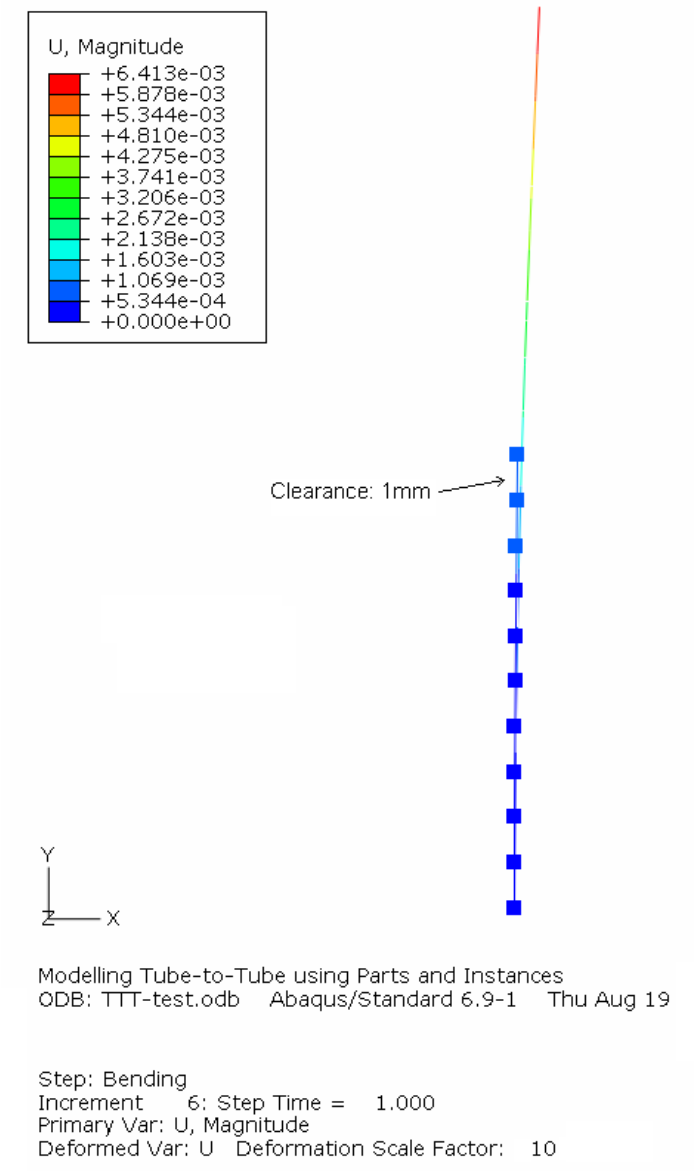

Figure 5. Deformation plot with 1 mm radial clearance

INPUT FILE

The inputfile shown below can be downloaded here (www.lhe.no/download/TTT-test.inp). An inputfile with opposite slide line (outer tube/pipe is slide line) can be downloaded here (www.lhe.no/download/TTT-test1.inp)

********** INPUT FILE STARTS HERE **********

```
*Heading
Modeling Tube-to-Tube using Parts and Instances
**
** August 19, 2010
** Created by Lars Hansen, L Hansen Engineering
** www.lhe.no
**
** PARTS
*Part, name=Beam-1m-P
*End Part
**
*Part, name=Beam-2m-P
*Node
      1,          0.,  0.0,          0.
      2,          0.,  0.1,          0.
      3,          0.,  0.2,          0.
      4,          0.,  0.3,          0.
      5,          0.,  0.4,          0.
      6,          0.,  0.5,          0.
      7,          0.,  0.6,          0.
      8,          0.,  0.7,          0.
      9,          0.,  0.8,          0.
     10,          0.,  0.9,          0.
     11,          0.,  1.0,          0.
     12,          0.,  1.1,          0.
     13,          0.,  1.2,          0.
     14,          0.,  1.3,          0.
     15,          0.,  1.4,          0.
     16,          0.,  1.5,          0.
     17,          0.,  1.6,          0.
     18,          0.,  1.7,          0.
     19,          0.,  1.8,          0.
     20,          0.,  1.9,          0.
     21,          0.,  2.0,          0.
*Element, type=B31
 1,  1,  2
 2,  2,  3
```

```
 3,  3,  4
 4,  4,  5
 5,  5,  6
 6,  6,  7
 7,  7,  8
 8,  8,  9
 9,  9, 10
10, 10, 11
11, 11, 12
12, 12, 13
13, 13, 14
14, 14, 15
15, 15, 16
16, 16, 17
17, 17, 18
18, 18, 19
19, 19, 20
20, 20, 21
*Nset, nset=_PickedSet2, internal, generate
  1,  21,   1
*Elset, elset=_PickedSet2, internal, generate
  1,  20,   1
** Section: Wellhead-20inx0_835in  Profile: Casing-20inx0835in
*Beam General Section, elset=_PickedSet2, poisson = 0.3, section=GENERAL
0.0324349, 0.0039912, 0.00282221, 0.0039912, 0.00798242
0.,0.,-1.
 2.1e+11, 8.07692e+10
*End Part
**
**
** ASSEMBLY
**
*Assembly, name=Assembly
**
**
*Instance, name=Beam-2m, part=Beam-2m-P
*End Instance
**
*Instance, name=Beam-1m, part=Beam-1m-P
*Node
      1,          0.,  0.0,          0.
      2,          0.,  0.1,          0.
      3,          0.,  0.2,          0.
      4,          0.,  0.3,          0.
      5,          0.,  0.4,          0.
      6,          0.,  0.5,          0.
```

```
      7,          0.,  0.6,          0.
      8,          0.,  0.7,          0.
      9,          0.,  0.8,          0.
     10,          0.,  0.9,          0.
     11,          0.,  1.0,          0.
*Element, type=B31
 1,  1,  2
 2,  2,  3
 3,  3,  4
 4,  4,  5
 5,  5,  6
 6,  6,  7
 7,  7,  8
 8,  8,  9
 9,  9, 10
10, 10, 11
*Element, type=ITT21, ELSET=TTT
11,  1
12,  2
13,  3
14,  4
15,  5
16,  6
17,  7
18,  8
19,  9
20, 10
21, 11
*Nset, nset=_PickedSet3, internal, generate
  1,  11,   1
*Elset, elset=_PickedSet3, internal, generate
 1, 10,  1
** Section: Conductor-30inx1in  Profile: Casing-30inx1in
*Beam General Section, elset=_PickedSet3, poisson = 0.3, section=GENERAL
0.0587781, 0.0039912, 0.00282221, 0.0039912, 0.00798242
0.,0.,-1.
 2.1e+11, 8.07692e+10
*INTERFACE, ELSET=TTT
0.001
*End Instance
**
*SLIDE LINE, ELSET=TTT, TYPE=LINEAR
Beam-2m.1, Beam-2m.2, Beam-2m.3, Beam-2m.4, Beam-2m.5, Beam-2m.6,
Beam-2m.7, Beam-2m.8,
Beam-2m.9, Beam-2m.10, Beam-2m.11
**
```

```
*Elset, elset=TTT, instance=Beam-1m, generate
 11,21, 1
*Nset, nset=Beam-2m-bottom, internal, instance=Beam-2m
 1,
*Nset, nset=Beam-1m-bottom, internal, instance=Beam-1m
 1,
*Nset, nset=Beam-2m-top, internal, instance=Beam-2m
 21,
*End Assembly
**
** MATERIALS
**
*Material, name=Steel
*Density
7810.,
*Elastic
 2.1e+11, 0.3
** ----------------------------------------------------------------
**
** STEP: Bending
**
*Step, name=Bending, nlgeom=YES
*Static
0.1, 1., 1e-05, 1.
**
** BOUNDARY CONDITIONS
**
*Boundary
Beam-1m-bottom, ENCASTRE
Beam-2m-bottom, ENCASTRE
**
** LOADS
**
*Cload
Beam-2m-top, 1, 1000000.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
*End Step
```