Brief Summary

1. Background Information

During offshore installation of a flexible pipeline, an abnormal wound pipeline at KP 2.0 was observed, as shown in below sketches.
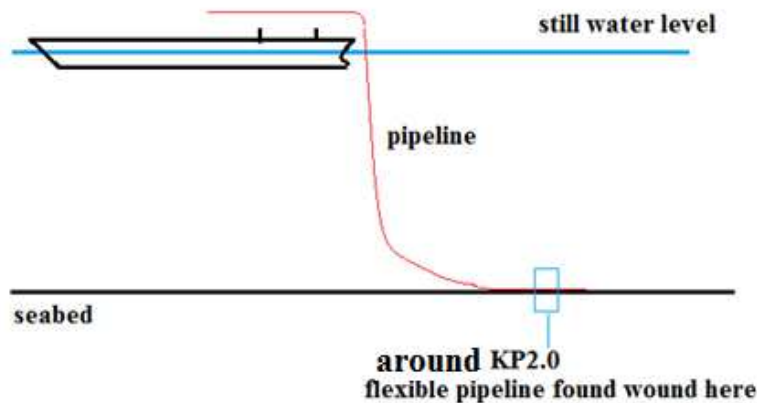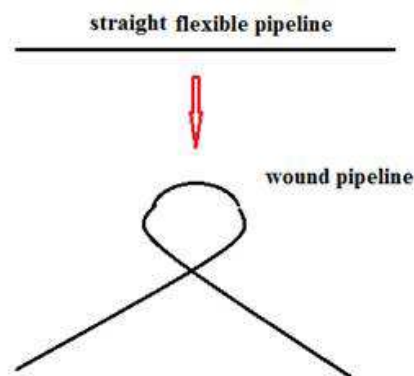

Figure 1

Figure 2 (diameter of such circle around 12m)

2. Purpose

The FE analysis is to simulate the deformation of wound pipeline.

3. Model

Please see the attached model files.

4. Methodology

4.1 Analysis Technique

Both ABAQUS/STATDARD and ABAQUS/EXPLICIT were used for such analyses. Unfortunately, the aforementioned deformation was successfully simulated.

4.2 Multiple Analysis Steps

According to my understanding, the following analysis steps were defined.

Step	Description	Loads
1	Assume that the pipeline is laid on the seabed; Contact pair defined for pipeline and seabed	Gravity Buoyancy (equivalent O.D. and w.t. of the

Step	Description	Loads
		pipeline is assumed based on rough estimation)
2	Lift the pipeline in order to keep the portion of un-laid pipeline on the barge	Prescribed displacement (on global z direction 122.45 m, i.e. the global position of tensioners on the barge)
3	Apply "actual" tension on the tip of pipeline	Concentrated force
4	… …	… …