Piping Flexibility Studies Using IDEAS Finite Element Analysis Software

Pedro Bergés, Summer Internships in Science and Technology – Fermi National Accelerator Lab
University of Texas at El Paso, Mechanical and Industrial Engineering Department
July 28, 2002

Abstract

The method developed in IDEAS to analyze the piping system to be used for the Main Injector Neutrino Oscillation Search (MINOS) experiment and the results obtained with it are presented. This paper includes a thorough description of the finite element (FE) model generated to analyze the system. It also describes the approach to the problem that the author found to be most efficient. Ultimately, results on the MINOS piping system are presented; giving the reader an idea of the kind of results and accuracy that this analysis generates.

This paper has two objectives. The first is to serve as an engineering note to make suggestions and changes to assure that the piping system will work correctly under the influence of temperature change. The second is to be helpful in future analysis of this sort and other kinds of structural analysis.

1 Introduction

The mechanical support piping system that was installed for the NuMI/MINOS project[1] will operate at a temperature higher than the temperature at which it will be installed. Therefore, the extraction enclosure part of the system will expand due a differential temperature of 40 deg. F. The rest of the system will suffer a differential temperature of 50 deg. F. Given that the piping system has large dimensions, proper precautions have to be made to avoid undesirable stresses and displacements. These could cause problems such as leaks, deformation, and component failure.

The purpose of this paper is to make a thorough analysis of the system. First, the analysis method is presented. Then, the approach that the author found to be the best for boundary condition generation is described. Finally, based on the results of this analysis, conclusions are reached and recommendations issued.

2 Analysis Description

2.1 Finite Elements

Although hand calculations could be appropriate for this system, the Finite Elements (FE) method was chosen. FE analysis (FEA) provides more flexibility in evaluating different boundary conditions and displaying the results. Roughly, FE modeling consists of generating a series of points called "nodes" that connect with other nodes to form elements. Then the computer solves the equations that apply for each element and displays the results. IDEAS is the computer-aided design (CAD) and FE software used for this analysis.

2.2 Generating the Geometry

Before nodes and elements can be generated, there needs to be geometry to be used. First, a 3-D model of the system was generated using extruded and revolved sections. For the reasons mentioned in the next section, beam meshing was used. Beam meshing can only be generated along edges or lines, not extruded sections. Therefore, the 3-D model was only used to trace 3-D lines and fillets along the pipe centerlines. In other parts of the system, the 3-D lines were created directly without first using extruded sections. Extruded sections are easier to visualize and may be first used for complicated systems. However it is much faster to generate the 3-D directly. In both cases, it is very important to first establish a coordinate system to follow throughout the drawing process.

Very useful commands for 3-D line generation are "on curve", when starting from an extruded section 3-D model. When 3-D lines are generated directly, "point to point" should be used, then in the pop-up menu the translated option should be selected for the second point of the line. This is more useful

[1] Drawing numbers 8875.115-ME-363561_01, 8875.115-ME-363561_02, 8875.115-ME-363487-01, 8875.115-ME-363487-02, NuMI Outfitting 6-7-6 (PP-4,PP-3, C-1, PP-11)

than typing in coordinates. Standard 2-D filleting may be used with 3-D lines to create the elbows. "Divide at" should be used to space fillets (elbows) and straight 3-D lines; the spacing used for this model is 1". The reason for doing this will be mentioned later.

An FE model can only be generated on a named part. When a series of 3-D lines are named as a part, it is not possible to add more lines to the part. Thus, expanding the model requires the creation of an assembly with the old part (3-D lines) and the new additions. Then, the command "join all" creates a new part that includes the old part and the new addition. It is then possible to make a new FE study on the new part. To avoid doing this, it is necessary to create the entire geometry of the model before meshing the lines.

2.2 Meshing

Meshing is the FEA step where nodes and elements are generated. In IDEAS, there are several methods to mesh a part. One of them consists of meshing a solid with solid elements. This is precise for analyzing compact and complicated parts. However, it may become computationally expensive when analyzing large models like the MINOS piping system for it creates a large amount of elements. Increasing the element's size decreases the number of elements, but this leads to solution errors. Shell meshing also creates many elements and leads to unnecessary complications when dealing with temperature. Thus it is not appropriate for this kind of analysis.

Beam meshing is the best technique for this kind of analysis. It generates a fairly small amount of elements and makes it possible to look at stress on parts of the system in more detail if desired. With this system, one node was generated every 10" of straight pipe and every ½" of curved pipe (elbows). The element lengths used with this model gave good results. The element lengths could probably be longer and still yield correct results, but it is easier to visualize the results of models with small lengths. Beam meshing made the FE model considerably more compact and easy to handle than if solid or shell elements were used.

Even though the system was meshed altogether, the software took the elbows and the straight pipe sections as separate. Constraint elements were first used to join the elbows with the straight pipe sections. However, two solution methods that were supposed to give the same results gave very different results. Therefore, this meshing method was incorrect. Other meshing methods were also attempted. Curved pipe elements offered an elegant way of modeling the elbows, but it was sometimes impossible to give them the correct orientation. Making and then meshing a continuous curve did not solve the problem. The best way of joining the system is to create a straight beam element between the straight pipe and the elbow. A beam element has to be created between two separate nodes. To do this, the last node of the straight pipe and the first node of the elbow were separated by 1"(this is where the "divide at" command becomes useful when making the geometry). Figures 1 and 2 illustrate this.

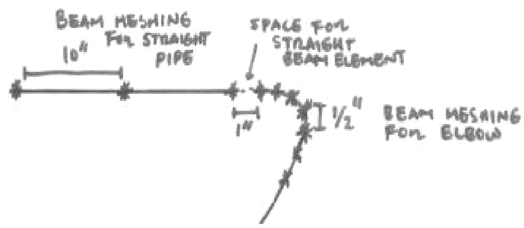


Figure 1. Elbow and straight beam meshing

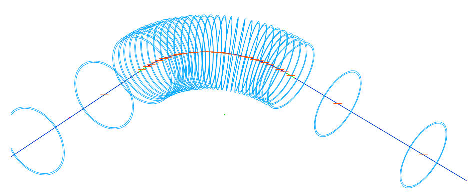


Figure 2. Elbow and straight beam meshing as seen on the screen

Apart from the different attempts to mesh an elbow with different techniques, the accuracy of the results obtained with each kind of mesh was compared with hand calculations (attachment 7). The curved pipe element was found to be the most accurate. However, the difference of the stress results obtained with it to the ones obtained with the mesh described above is only 8%. This makes the stress results reliable as long as they are not very close to the allowable stress.

To model the Tees of the system two of the approaching lines are spaced by 1" to the other line. Then, two straight beam elements are created. This is very similar to what was done with the elbow. Figure 3 illustrates this.

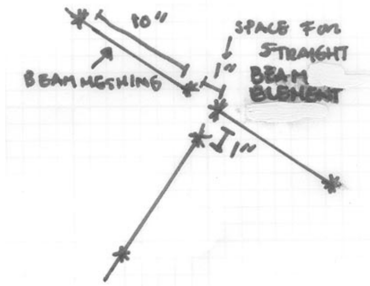


Figure 3. Tee meshing

There are parts of the system where concentric reducers are used. It was attempted to model the concentric reducer with solid meshing, and then using constraint elements to join it with the pipe. This proved feasible but computationally expensive and unnecessary for this kind of analysis. It is much better to model the reducers as a tapered beam element

2.3 Boundary Conditions

To simplify the analysis the small lines that branch out from the pipe are not considered. The FEA was done on one of the pipes (return), for the supply and return pipes are almost identical.

A temperature of 110 deg F was applied on the nodes of the extraction enclosure part of the system. Also, a temperature of 100 deg F was applied on the nodes on the carrier tunnel. Hence, generating temperature sets where the reference temperature was entered as 70 deg F. This condition simulated a pipe that was installed at a temperature of 70 deg F and operated at temperatures 40 and 50 deg F higher.

All boundary conditions are generated on nodes. The displacement restraints create displacements on nodes. Nodes can be either forced to move or left to move freely in the six nodal degrees of freedom (DOFs), x, y, and z rotations and displacements. The nodal DOFs are based on the active coordinate system. It is possible to create and associate other coordinate systems with the part to make better restraints. For example, such a restraint that allows longitudinal translation of a pipe that is not parallel to x, y or z of the default coordinate system.

It is important to stress the fact that when a restraint is specified, the "amplitude" of a fixed motion is how much a node is forced to move. For instance, if a value of 1 is input in the displacement in the x direction box, it will not allow it to move from 0 to 1 nor from –1 to 1. It will force the node to move 1 in the +X direction. Hence this boundary condition does not allow the part to move as pipes do in real life. Figure 4 shows the strut hanger that is used to support pipe in the system. The supports are a size bigger so they allow at least ½" of side movement.

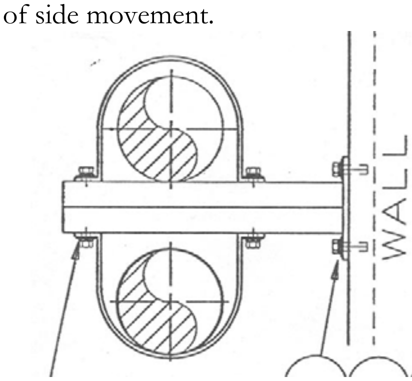


Figure 4. Strut hangers.

The way pipes are supported sets the problem of finding a boundary condition that allows movement before coming into action. The solution is not itself in a boundary condition, but in the meshing part of the software. Node to ground "gap elements" are appropriate for this problem. They create a space between a node and an obstacle. The contact direction can either be positive or negative x, y, or z (here the creation of new coordinate systems may be handy as well). The properties of a gap element can be edited in the element property tables. Once the model is solved, the software iterates to find were the pipe actually makes contact with the supports. IDEAS also computes the magnitude of the contact force.

2.4 Approach to the Model Solution

Once the model is meshed, simple boundary conditions should be applied to check for errors. These can be as simple as clamping all the ends of the system. The most common error is to have a discontinuity in the pipe, which may be caused by forgetting to create an element between an elbow and straight pipe. When this happens, the pipe will look "broken" when checking the displacements in the post-processing part of the software. To correct the mesh, it is necessary to delete the results. When the mesh is corrected, the model is ready for further tests.

In IDEAS, it is possible to make different boundary condition sets for the same FE model. A boundary condition set contains all the loads and restrains of the system.

It is tempting to create many different boundary conditions for the same FE model. However, this approach may only be useful for simple systems for it may become very complex for larger systems. However, it is recommended to use it to check for specific conditions in either small or large models, such as putting a clamp on the system or supporting the small pipe sections too tightly.

Before beginning to describe the approach to the model solution, it is important to mention that the model should be as large as possible. Adding parts to it is possible but time consuming (as described in generating the geometry), since the model has to be meshed again.

The approach that the author found to be the most useful is the following. First, apply the most basic boundary conditions. Then, the lateral displacements of the pipe must be observed in to determine were the motion tolerances are exceeded (when the strut supports comes into play). This is done in the post processing part of the software by using the contour diagrams and the report writer.

Where the lateral pipe displacement tolerances are exceeded, a minimum amount of gap elements should be applied in the direction of the contact (+x, -x, _y, -y, +z, and -z). To add gap elements, it is necessary to delete the results. Gap elements are part of the model's mesh; the mesh cannot be modified while results for that particular mesh exist. Once more gap elements are added, the FE model has to be solved again, then the lateral displacements checked. If the tolerances are exceeded the process has to begin once again [add more gap elements]. Only one gap element may be connected per node.

To view the magnitude of the contact forces in post-processing, it is necessary to store the constraint forces in the "output selection" before solving the model.

When no part of the model exceeds the support lateral motion tolerances the model is solved. It is now time to get detailed results for the model. Possible results are restraint reaction vectors, nodal displacements, and contact forces. All this is done with the "report writer" by selecting the elements or nodes of interest. Stress results are better viewed with a contour diagram.

3. Testing the Piping System

3.1 General System Description

The total length of the pipe in the system is about 2000 feet. The models of the system's parts are very detailed. This and the results can be observed in appendixes at the end of this paper. Figures 5 shows the extent of the system. Figure 6 shows the amount of detail of the FE model.

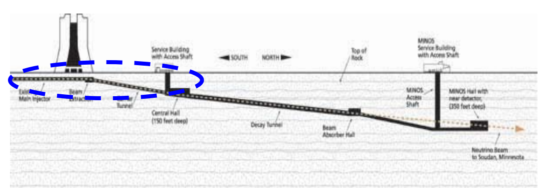


Figure 5. The blue oval shows the approximate area for LCW piping

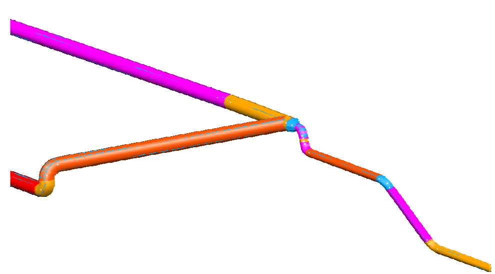


Figure 6. Snapshot of a part of the system in the extraction enclosure near Quad Q613

The system is clamped at the parts where true rigid conditions exist, such as the connection with a pump or a heart exchanger, or a specific clamp. As it was mentioned before, gap elements are only added when the lateral displacements are greater than those allowed by the supports.

There are three parts of the system. One of them is already installed. There is a rough sketch for the second one. The third part has not yet been designed. The following sections describe the details of the system's parts and present the results.

3.2 Part 1, MI-62 – Extraction Enclosure

3.2.1 Description

This part of the system already installed and goes from the MI-62 building to the extraction enclosure. Here the system gets divided into two branches with a Tee and changes diameter 4 times. This part of the system uses 4", 6", 3", and 2" schedule 10 pipe. Figure 6 shows how the system looks at the Tee location. The temperature change for this system is expected to be only 40 deg F. Since the tunnel had warmed to 80 deg F before all of this piping was installed

The system is clamped at the left end and it is left free to expand longitudinally at the other two ends. The geometry in the MI-62 building (after the pipe goes up the shaft) does not have the correct orientation. However, this is not I important since only a moment about z is transmitted through the pipe that goes up the shaft. The reaction vector (x, y, and z forces and moments) on one of the free ends is calculated in case that end is clamped. For a more detailed diagram of the results go to attachments 1, 2, and 3 at the end of this paper.

3.2.2 Results

Since the approach described in section 2.4 was followed to solve the model, the lateral displacements are all within the ½" allowable. Here is a list of the most important results.

- Maximum stress **1570 psi** (Von Misses)
- Reaction vector at clamped end {Fx, Fy, Fz, Mx, My, Mz} in pounds and inch-pounds. **Expected(exp) {-7, -3, -1.5, -92, -596, 1292}**
- Maximum contact force **0#**
- Longitudinal translation in the "right end" (towards the Main Injector) **exp +2.35", wcs(worse case scenario) +3.71"**
- Longitudinal translation of the left end **exp +.44", wcs +1.75"**
- Reaction vector if the left end is clamped. **exp {15, 1, 0, 55, 86, 1045}, wcs {-20, 1, 0, 70, 10, 1344}**

3.2.3 Recommendations

The system does not require any substantial changes. The possibility of cutting the 4" pipe that comes from the MI62 building to add an expansion loop was explored and proved useless. This is because the shaft and the turns below the MI62 building provide unsuspected flexibility. A field inspection will ensure that the following recommendations are observed

- The lines that connect the pipe with the magnets on the main injector should be able to take the 3.7" expansion of the pipe. If the material is flexible hose it should be able to expand to or have the following length.
  L = (M^2 + 3.7^2)^.5 + 2
  (L = (M^2 + 2.35^2)^.5 + 2 **if** the boundary condition enclosed with a red rectangle in attachment 1 is satisfied)
  Where L is the length of the hose and M is the length from the pipe to the magnet.
- The smaller straight pipe section that branches out of the Tee near Quad should be allowed to expand 2.5" (wcs) towards the main injector Q601 (the opposite direction to MI-62).
  (1.5" of clearance is OK **if** the boundary condition enclosed with a red rectangle in attachment 1 is satisfied)
- The turn just below the MI-62 building should be free to move 1.2". It is enclosed on a red circle in attachment 1.
- The heat exchanger and the pump in the MI-62 building should be able to take the following reaction vector **{-7, -3, -1.5, -92, -596, 1292}**

3.3 Part 2, Carrier Tunnel – Target Hall

3.3.1 Description

Attachment 4 at the end of this paper shows a picture of this part of the system, which is actually larger than part 1. The pipe runs down the carrier tunnel where after a few turns it gets divided with a Tee and changes diameter from 4" to 3". One end remains horizontal to cool electrical equipment and the other goes up the shaft to the target hall service building. The temperature change for this system is expected to be 50 deg F; because this portion of the tunnel is not likely to be appreciably heated above 60 deg F. before the pipe is installed.

The system is clamped at the end of the long straight-pipe spans, the carrier tunnel and the shaft to the target hall service building. These long straight pipe spans have expansion loops. For a more detailed diagram of the results and geometry go to attachments 4, 5, and 6 at the end of this paper.

3.3.2 Results

The approach described in section 2.4 was also followed this model. Thus, the allowable lateral displacement of ½" is not exceeded. Here is a list of the most important results.

- Maximum stress **20100 psi** (Von Misses) occurs at the carrier tunnel expansion loops. This value is below the allowable 28150 psi [1]
- Highest reaction vector at clamps (see attachment 4 for the other clamps) **wcs {-3225, -47, -509, -434, 0, -2748} Ft = 3270#** occurs at the left-most clamp, at the top of the carrier tunnel.
- Maximum contact force **73# (out of the carrier tunnel wall)**
- Longitudinal translation of the "left end **exp +.44"**
- Longitudinal translation of the "right end" **exp +2.15" (if clamp is removed)**

3.3.3 Recommendations

This part of the system seems to be OK as it is. However the stresses and displacements on the carrier tunnel expansion loops seem to be unnecessarily high. The next part of the system is to be designed; the possibility of removing the clamp in the upper part of the carrier tunnel will be explored with the design of the part 3.

- ❑ Make sure the clamps of the system can provide the maximum reaction vector **{-3225, -47, -509, -434, 0, -2748} plus** the weight of the pipe.
- ❑ Make sure the supports in the carrier tunnel and the pipe section marked with a red circle in attachment 4 can provide a lateral force of 73#.
- ❑ Support the member marked with a red circle in attachment 4 with **only one** strut support at the middle of it.
- ❑ The components at the left end of the system should be capable of bearing a ½" longitudinal expansion in the –x direction.
- ❑ The strut supports on the carrier tunnel should be at least 10" before the elbows of the expansion loops. The expansion loops themselves should not have any supports. The same applies to the expansion loops in the shaft.

3.4 Part 3, Extractor Enclosure

3.4.1 Description

A rough sketch of this part had to be designed and tested. The design of this section has the main objective of not contributing, rather alleviating, the stresses and displacements of the other two parts of the system. This part connects with part 1 at **A** (labeled with a circled red A in the attachments) and with part 2 at **B** (labeled with a circled red B in the attachments). The coordinate system for all three parts is the same.

Attachment 8 shows the dimensions and the results of the piping designed for the extractor enclosure. This design, however, should be used **only if** the downstream piping is clamped or fully restrained at **B**. This FE model is clamped at **B** and the wcs force and moment vector is applied to end **A**. According to Newton's 3rd law, the wcs force and moment vector on part 3 **A** is the opposite of the wcs reaction vector on part 1 clamp at **A**. This approach is not exact but conservative enough to prove that the design is appropriate.

It is important to mention that the piping in this part is located on the x-z plane. That is, there is no pipe along the y-axis. The drawing in attachment 8 may be superimposed to drawing 8875.119-ME-406291 for a better reference.

3.4.2 Results

The approach described in section 2.4 was also followed for the solution of this model lateral displacements do not exceed the ½" allowable. Part 1

- Maximum stress **589 psi** (Von Misses)
- Displacement of the "left end" **A** (the one that connects with part 1). **Dx=1.20", Dy=-0.52, Dz=-0.50"**
- Reaction Vector at **B {-20, -1, -2, 278, -2445, -442}**
- Total reaction vector at clamp **B**. Required reaction for the pipe in the extractor enclosure (part 3) plus the required reaction for the carrier tunnel.
  {-20, -1, -2, 278, -2445, -442} +
  {-3225, -47, -509, -434, 0, -2748} =
  {-3245, -48, -511, -156, -2445, 3190}
  Total force 3285 #

3.4.3 Recommendations

- A minimum support clearance of ½" is required. This will allow the pipe to have some lateral displacement.
- Do not support the 96" vertical pipe segment tightly. Allow at least ½" of lateral displacement.
- Make sure the clamp at **B** is strong enough to provide this reaction vector **{-3245, -48, -511, -156, -2445, 3190}** where the forces are in pounds and the moments in inch-pounds.

4 Acknowledgements

This has been my second great summer on Fermilab. It has been an honor to work again with my supervisor David Pushka. I would like to thank Dianne Engram, Dr. McCrory and Dr. Davenport for their support, as well as my sponsors of the REU program back in El Paso. The skills I have acquired and the experiences I've lived by doing research are very significant to me.

5 References

[1] Piping Engineering. Sixth Edition, Tube Turns, 1986.

[2] Shigley, Joseph E. and Mischke, Charles R. Mechanical Engineering Design. Sixth Edition. McGraw Hill, 2001.

[3] Crocker, Reno C. and King, Sabin. Piping Handbook. Fifth Edition. McGraw Hill, 1973.

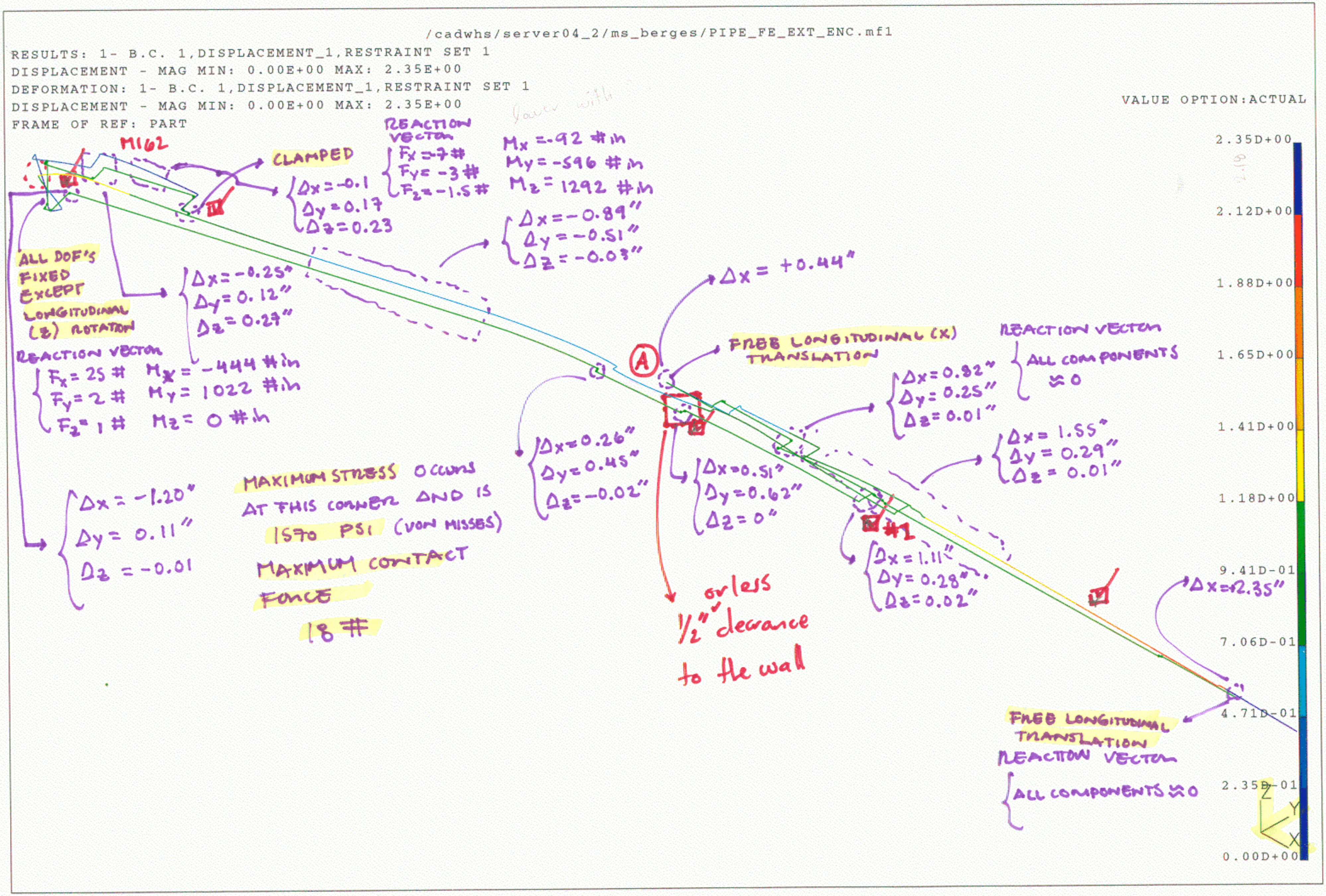
[4] IDEAS 8 online manual. Structural Dynamics Research Corporation, July 2001.

ATTACHMENT #1

MI-62 — EXTRACTION ENCLOSURE PIPING

FLEXIBILITY ANALYSIS USING A DIF. TEMPERATURE OF 40 °F

ATTACHMENT #1



Plotted by berges on 25-Jul-02 , File: ext_enc.pff

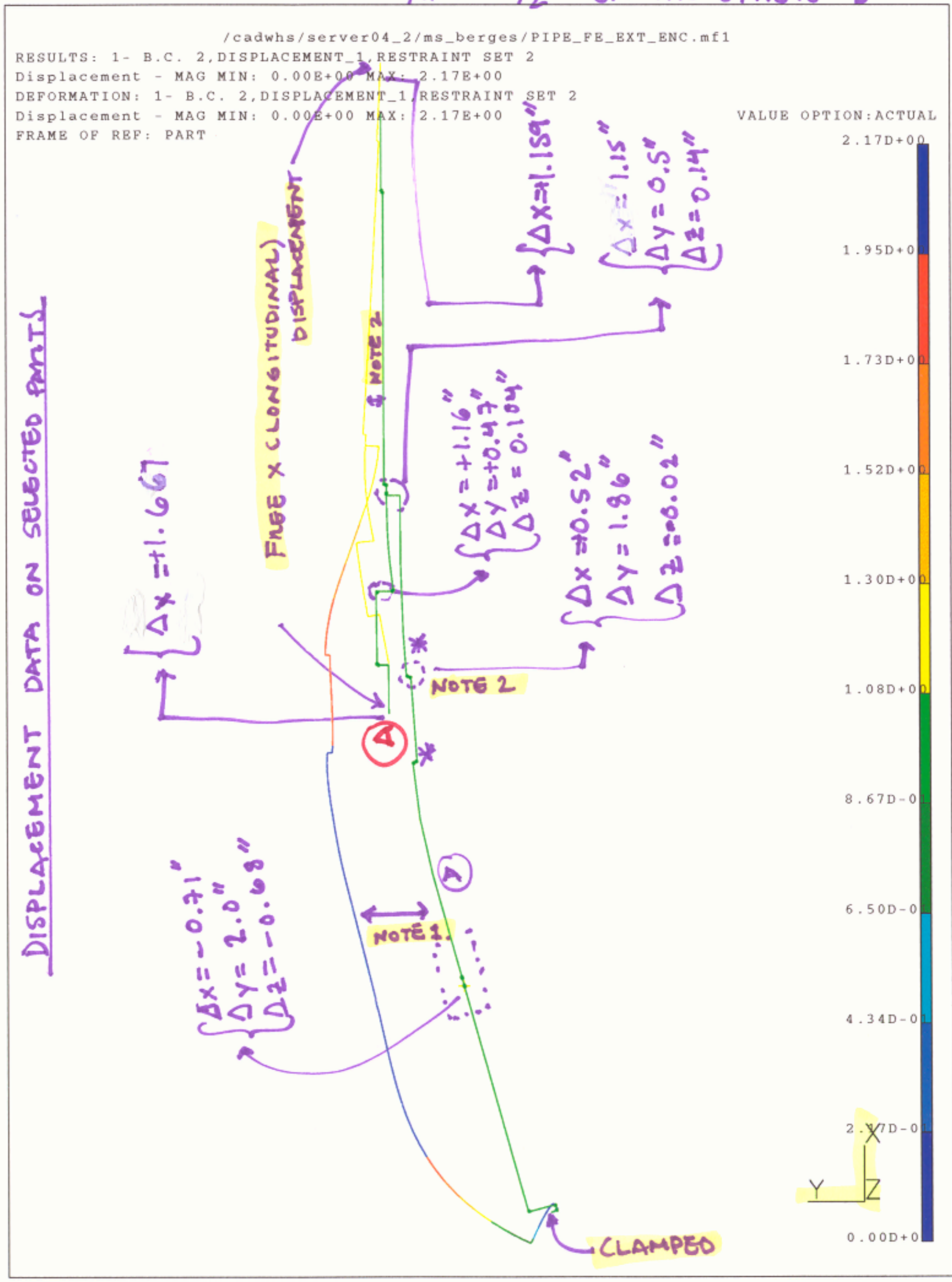
ATTACHMENT #2

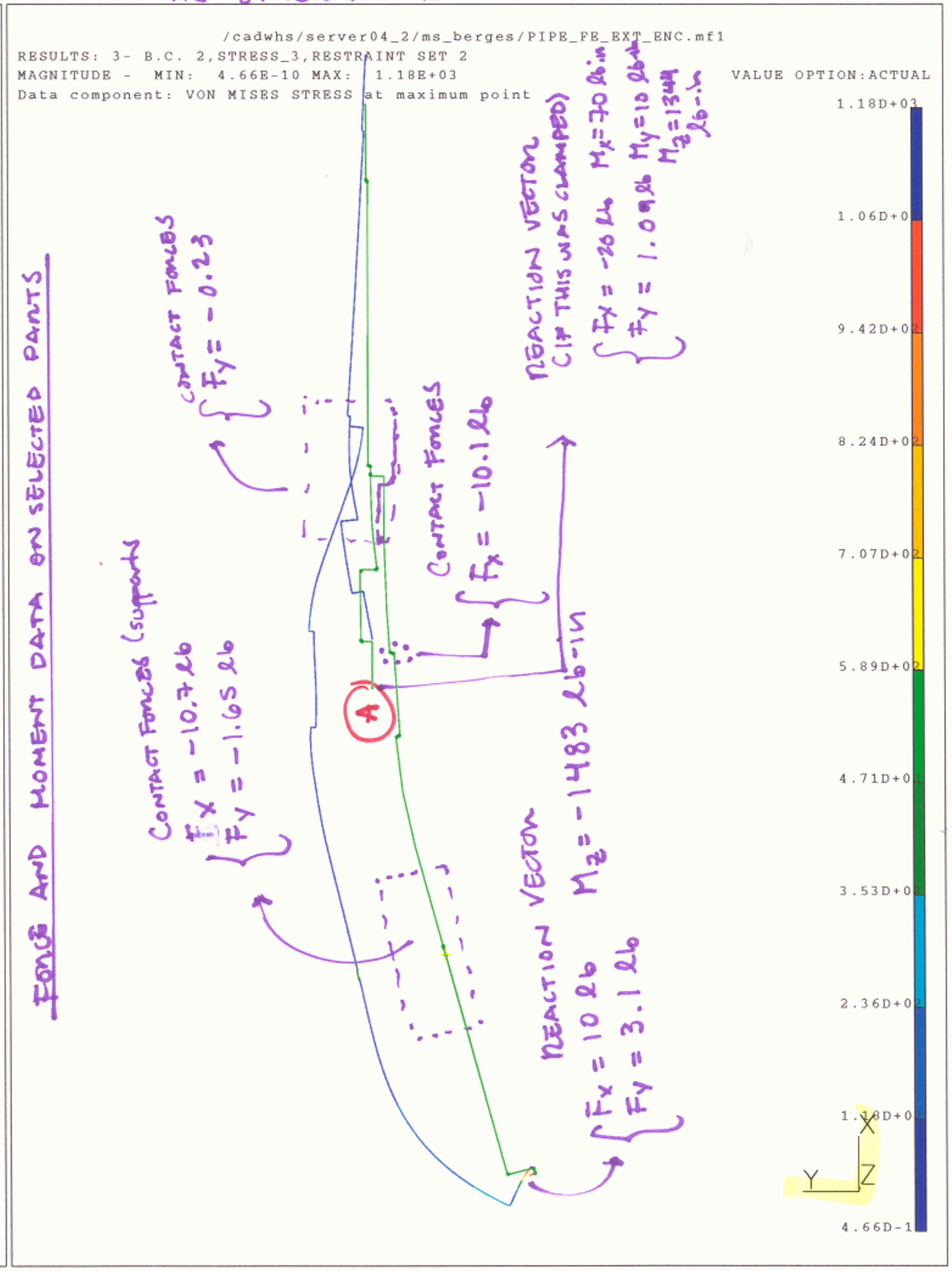
NOTE 2 THIS PIPE SEGMENT IS ALLOWED TO MOVE 1/2 IN THE +X DIRECTION

* 1/2" ALLOWED IN THE +X DIRECTION

NOTE 1 THIS MODEL ALLOWS A 2" DISPLACEMENT ON THE LONG BRANCH (A) ON THE +Y DIRECTION AND 1/2" ON THE OTHER BRANCHES IN THE OTHER TWO BRANCHES

ATTACHMENT #2

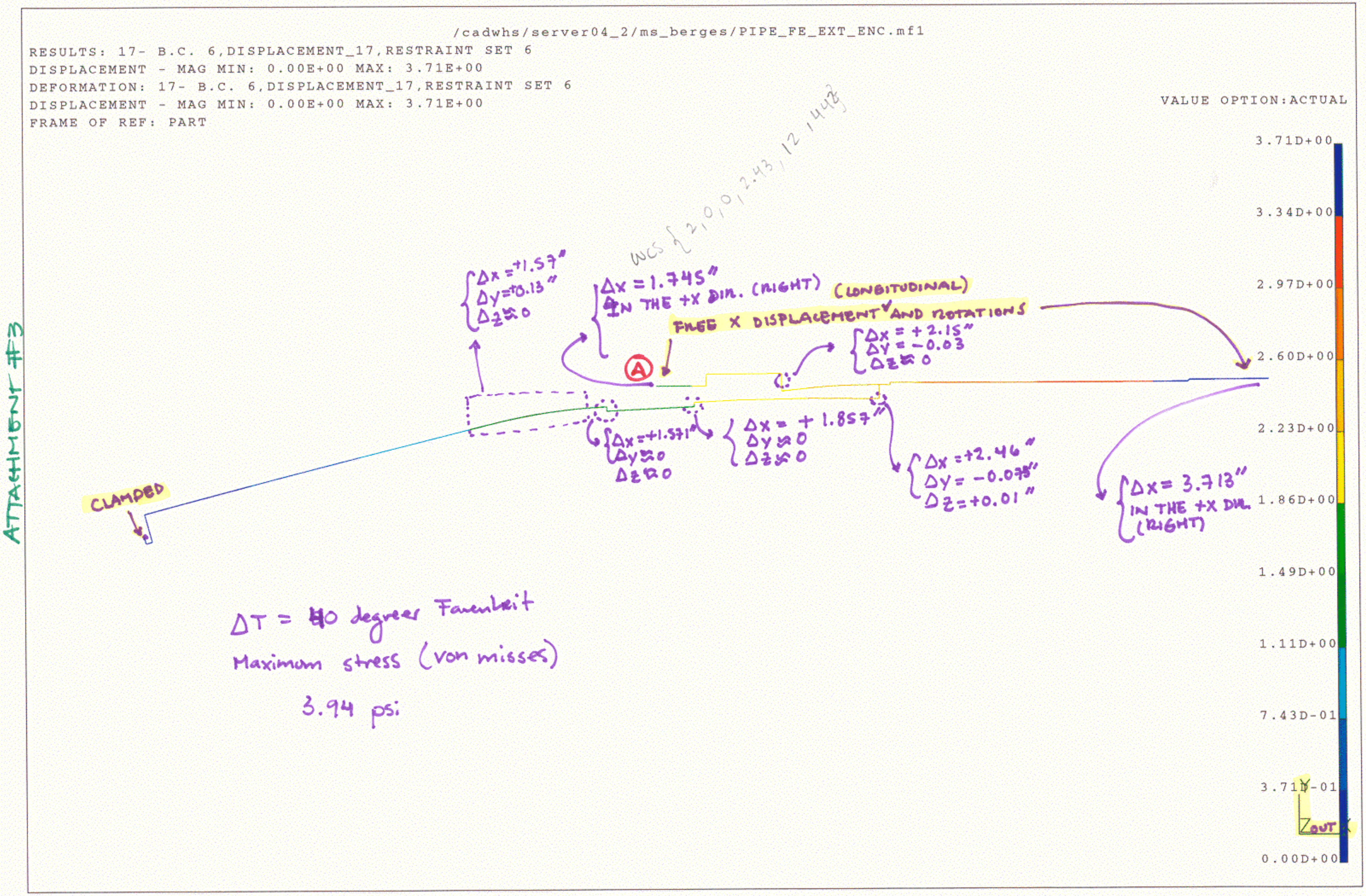




Plotted by berges on 19-Jul-02 , File: ext_gop_loop.pff

BOUNDARY CONDITIONS ARE HIGHLIGHTED

ATTACHMENT #3

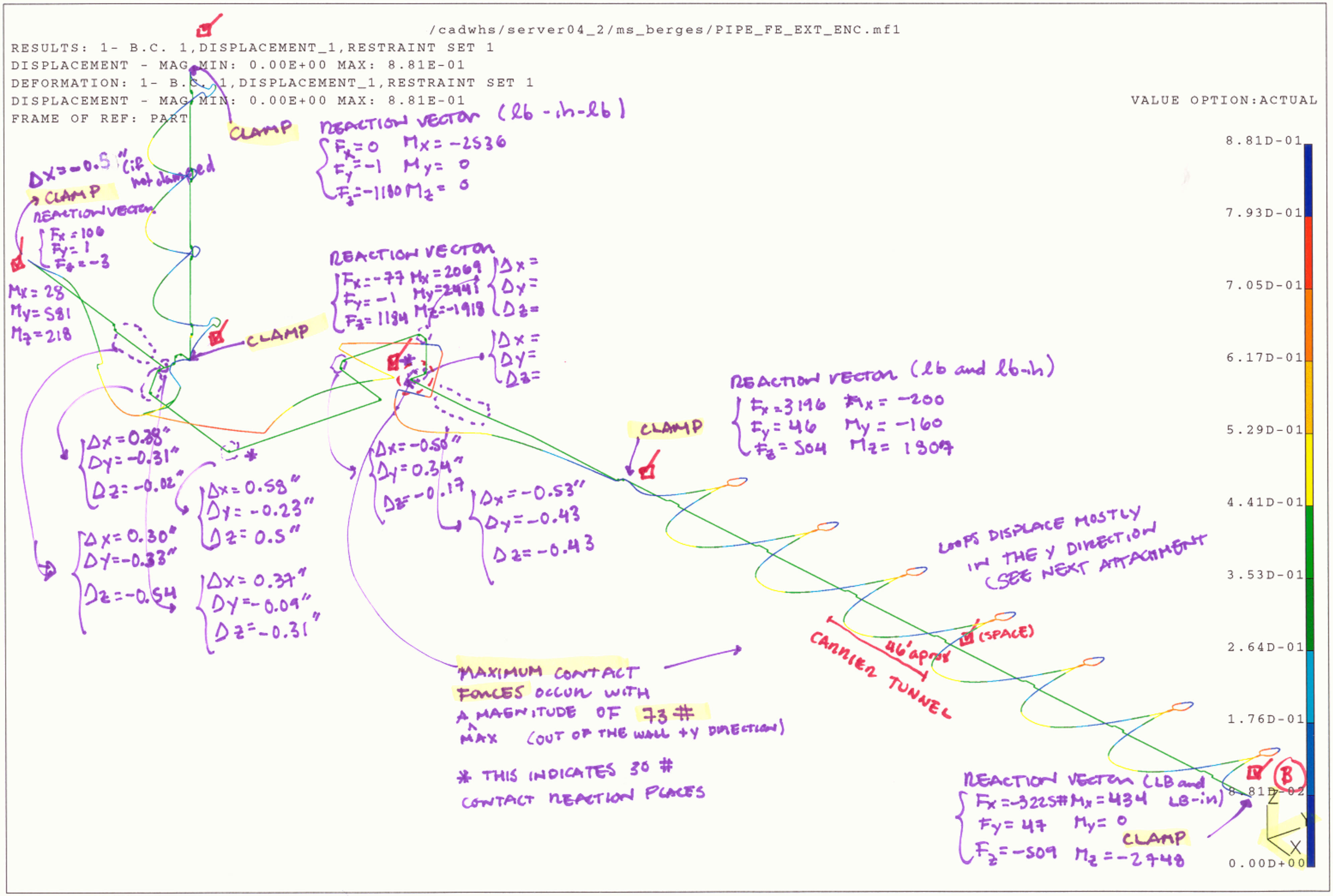


Plotted by berges on 10-Jul-02 , File: 6-d.pff

ATTACHMENT #4

CARRIER PIPING FLEXIBILITY ANALYSIS

ΔT= 50°F



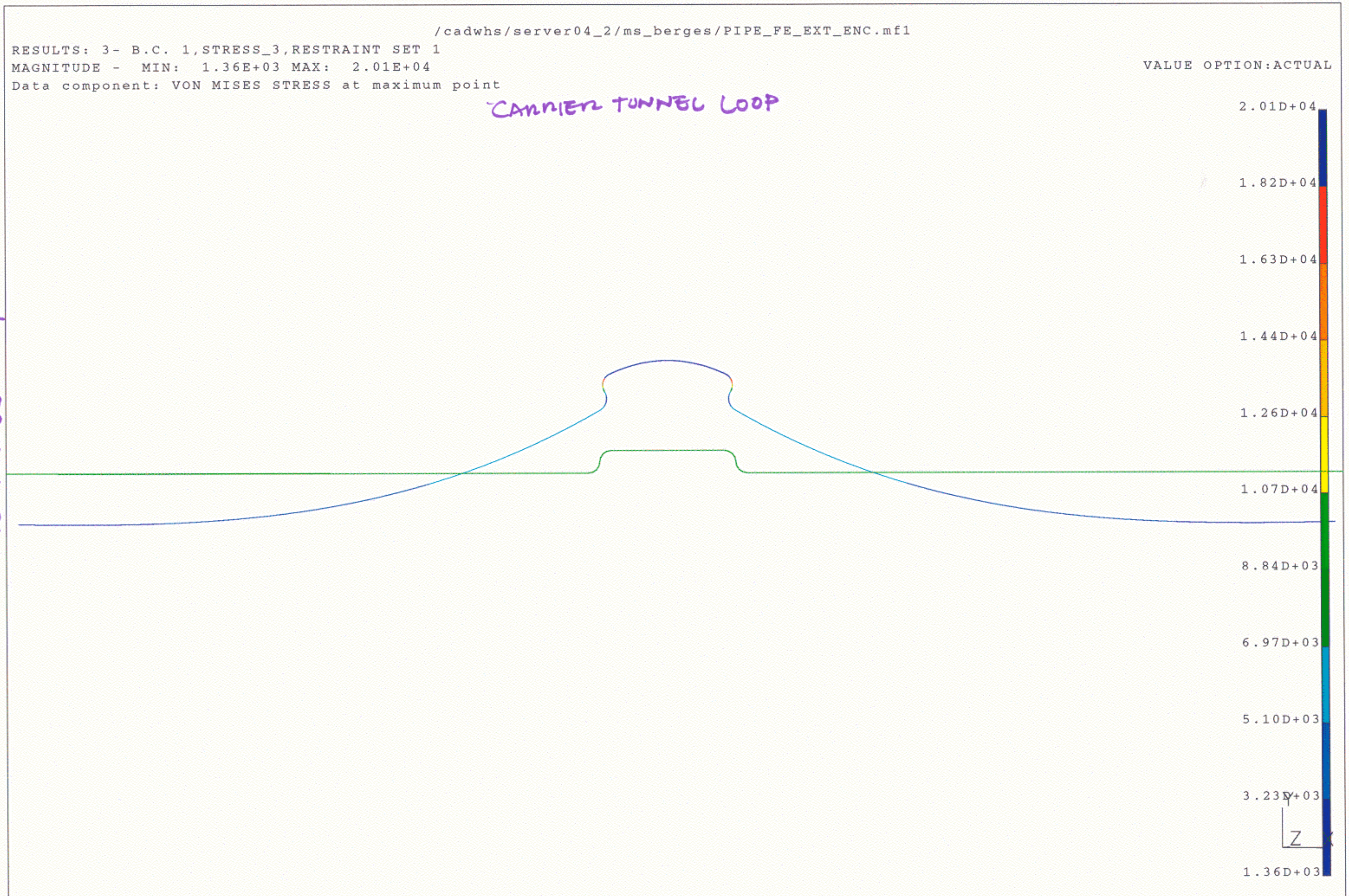
ATTACHMENT #4

ATTACHMENT 5

DETAIL FROM ATTACHMENT 4

ATTACHMENT 5

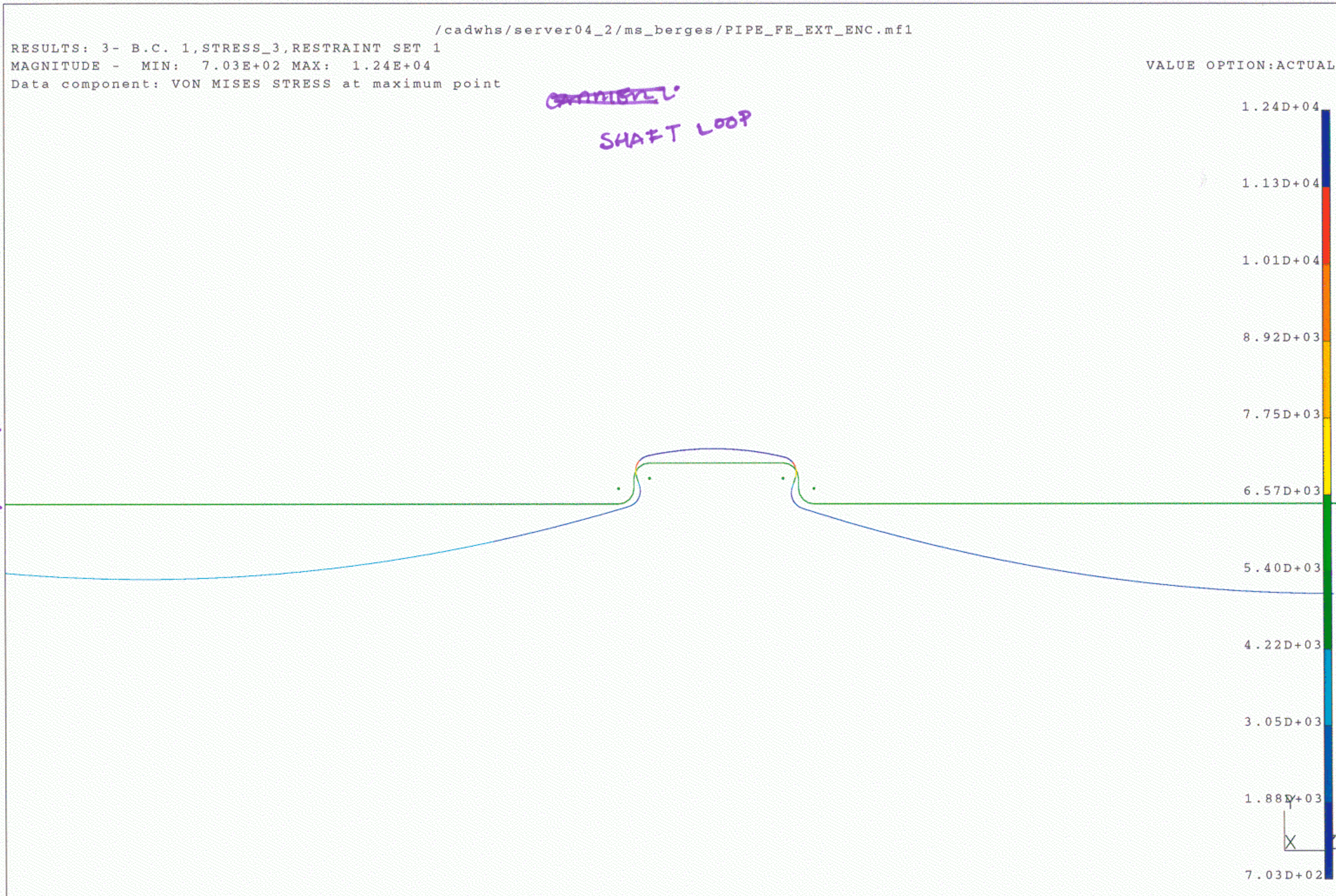
CARRIER TUNNEL LOOP



Plotted by berges on 26-Jul-02 , File: carrier-loop.pff

ATTACHMENT 6

DETAIL FROM ATTACHMENT 4



ATTACHMENT 6

SHAFT LOOP

Plotted by berges on 26-Jul-02 , File: shaft-loop.pff

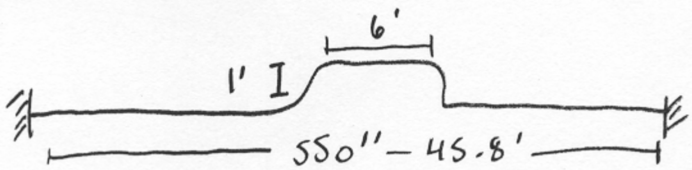
ATTACHMENT 7

FERMILAB ENGINEERING NOTE	SECTION	PROJECT	SERIAL-CATEGORY	PAGE

SUBJECT	NAME	
HAND CALCULATION FOR STRESSES IN CARRIER PIPE LOOP	DATE	REVISION DATE

ATTACHMENT 7

THE THERMAL EXPANSION LOOPS IN THE CARRIER PIPE ARE OVER-STRESSED, AS IT WAS PROVED WITH FEA. A HAND CALCULATION OFF THE STRESS IN THE LOOP WILL BE MADE TO PROVE THAT THE LOOPS ARE NOT ADEQUATE



THE PROCEDURES OUTLINED IN THE PIPING ENGINEERING 6TH EDITION BY TUBE TURNS (1986) WILL BE USED

$U = 45.8'$

$H = 1'$

$W = 6'$

$W/U = 0.131$

$H/U = 0.0218$

$f_s = 14$

$n = 7.5$

$U + 2H = L_s = 47.8'$

$n \cdot L_r = 87'$

$L_s + nL_r = L = 134.8'$

$L/D_r = f_1 = 9.38$

$\Delta T = 50°F$

$f_e = 12970$

L_r from table 1 $= 11.6$

$D_r = 14.36$

$$\frac{U'}{U} \cdot \frac{f_e f_s}{f_1} = S_E = 19358 \text{ psi}$$

$S_A = 28150$

ATTACHMENT #8

ATTACHMENT #9

EXTRACTOR ENCLOSURE PIPING
TENTATIVE DESIGN FLEXIBILITY ANALYSIS