

IMPROVEMENT OF MODELING AND EVALUATION OF PIPING SYSTEMS IN NUCLEAR FACILITIES

P. Markov*, A. Musil*

Summary: *The paper presents the strategy of modeling and evaluation of piping systems in nuclear facilities in ANSYS computer code with use of Piping Models module. The module incorporates macros developed to automate the modeling of pipelines including macro for easy modeling of branch connections (tees) in accordance the provisions of ASME Boiler & Pressure Vessel Code, Section III, Division 1 – Subsection NB, Class 1 Components (1992). For the developed computational models with defined boundary conditions and loadings, nodal forces and moments were computed. Model geometry and results of analyses are exported into data files which are then imported into the evaluation program set up in Mathcad. Thus the program contains the complete information of the FEM model necessary to carry out automatic evaluation in which the stresses at all nodes of the model are determined following the guidelines of the code and subsequently, the obtained values are compared with the stress allowable values specified for the material of the piping.*

The presented procedure makes the development of a pipeline model significantly faster, automates the subsequent evaluation process, and minimizes the risk of human-factor error. It can be utilized for linear or nonlinear static and dynamic analyses including seismicity.

Besides the detailed description of the model development in ANSYS and the implementation of macros, the examples of analysis of the concrete piping system is also presented in the paper.

1. Introduction

Numbers of specialized program modules which can have variously close relation to general FEM systems or can be absolutely autonomous have been developed for modeling of piping systems. An example of such modeler is called Piping Models and is incorporated in ANSYS computer code. Such solution is very convenient since the computation models developed using Piping Models module are standard 3D computational models built-up from one-dimensional finite elements (PIPE16, PIPE18).

* Ing. Petr Markov, CSc., Ing. Aleš Musil: STEVENSON AND ASSOCIATES a.s., Kancelář v České republice; Vejprnická 56; CZ-318 00 Plzeň; tel.: +420 377 350 233, fax: +420 377 350 230; e-mail: markov@stevenson.cz, musil@stevenson.cz

During the development of piping systems for nuclear facilities designed and evaluated in accordance with the ASME BPVC Code (ASME Boiler & Pressure Vessel Code, 1992) it turned out that a branch connection modeled using standard procedure TEE provides the creation of either welding tee (WT) or unreinforced fabricated tee (UFT) as specified in the code. Nevertheless, the resulting computation model does not meet the code requirements in all aspects. Therefore, we created a macro which applies certain commands from Piping Models and eliminates this inadequacy. In case of a branch connection of UFT type it models the flexibility for both bending directions located on the surface of run pipe at the juncture of run pipe and branch pipe according to the guidelines of the code (1992).

Piping Models module enables comfortable modeling of straight sections, curved and mitered bends, valves, flanges, reducers, flexible supports, and supports with gap. In order to model branch connections macro *tee.mac* was developed. It enables to create the model of a branch connection of UFT or WT type into the chosen node of a pipeline, in a similar manner as standard TEE procedure which models branch connections simpler than the code (ASME Boiler & Pressure Vessel Code, 1992) requires.

The other macros which have been created are *beg.mac*, *rst.mac*, and *rse.mac*. Macro *beg.mac* serves for proper defining of all input parameters for modeling. Using macros *rst.mac*, *rdy.mac* or *rse.mac* the data needed for subsequent evaluation in accordance with the ASME BPVC code (1992) are saved after the performance of the computation analyses.

The resulting product of Piping Models module is a standard computational model of ANSYS code. After the specification of boundary conditions and static or dynamic loadings, nodal displacements, forces and moments are computed.

Outputs of the FEM model – model characteristics (model geometry, element types, real constants, etc.) and computed values of bending and torsional moments are listed in six data files which are imported into the Mathcad program for further evaluation.

2. Code requirements for modeling of branch connections

ASME BPVC Code (1992) requirements for branch connections of pipelines are specified in paragraphs NB-3686.4 “Welding Tee or Branch Connection” for WT type and NB-3686.5 “Branch Connections in Straight Pipe” for UFT type of tee connections.

The computation model of a branch connection of WT type has to meet the assumption that run pipe and branch pipe of the branch connection are extended to the intersection of their centerlines. This intersection is assumed to be rigid as well as the branch pipe segment between the centerline and the surface of the run pipe.

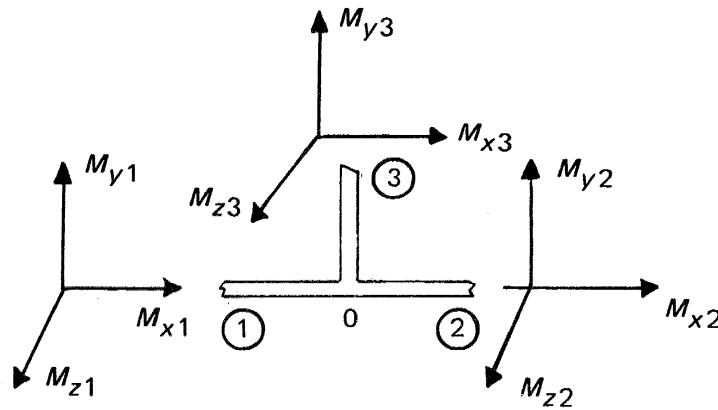


Figure 1 UFT type tee connection (from the code [1]) also with moments M_{x3} and M_{z3}

The basis of the computation model for UFT type of branch connection, which is shown in Figure 1, is identical with the model of WT type. In addition, it contains an element of negligible length on the surface of run pipe to represent local flexibility of branch pipe in bending both in the plane defined by the axes of run pipe and branch pipe and in the perpendicular plane. This flexibility must satisfy:

$$\Phi = kMd / EI_b \quad (1)$$

where:

- Φ – local rotational (turning) displacement of branch pipe
- k – flexibility of the wall of run pipe in a specified direction,
- M – bending moment in a specified direction,
- d – outside diameter of branch pipe,
- E – Young elasticity modulus,
- I_b – cross-sectional moment of inertia of branch pipe (calculated with d and T_b'),
- T_b' – nominal thickness of branch pipe defined for various types of reinforcement (paragraph NB-3643 of [1]).

The values of k_{x3} related to moment M_{x3} which acts perpendicularly to the plane defined by the axes of run pipe and branch pipe are defined as (see (ASME Boiler & Pressure Vessel Code, 1992), paragraph NB-3686.5):

$$k_{x3} = 0.1(D/T_r)^{1.5} [(T_r/t_n)(d/D)]^{1/2} (T_b'/T_r) \quad (2)$$

and the values of k_{z3} related to moment M_{z3} which acts in the plane defined by the axes of run pipe and branch pipe are calculated using:

$$k_{z3} = 0.2(D/T_r) [(T_r/t_n)(d/D)]^{1/2} (T_b'/T_r) \quad (3)$$

where:

D – outside diameter of run pipe,

d – outside diameter of branch pipe,

T_r – thickness of run pipe,

t_n – thickness of branch pipe with reinforcement according to paragraph NB-3686.5 of the code (ASME Boiler & Pressure Vessel Code, 1992),

T_b' – thickness of branch pipe according to Fig. NB-3643.3(a)-1, of the code [1].

3. Development of the computation model based on the requirements of ASME BPVC code

In case of 3D beam elements, the first requirement (rigid connection at the intersection of the run pipe and branch pipe centerlines) is satisfied automatically due to basic properties of beam elements.

The second and the third requirement were accomplished by programming using “ANSYS Parametric Design Language” (APDL). It was decided to follow the routine of “Piping Commands” (commands of Piping Models module) and applied some of its commands in macro *tee.mac*.

It was intended to minimize the number of input parameters in the macro. All data of geometry, material properties and real constants which contain dimensions of run pipe and branch pipe are read from the computational model in the state which it is at the moment of initiation of the macro.

After the reading of these data, TEE command (standard command of Piping Models module) is executed with the parameters which create three new pipe elements with shared node at the intersection of the run pipe and branch pipe centerlines. These new elements form one PIPE17 element. PIPE17 element is a combination of three uniaxial elastic straight pipe elements (PIPE16) arranged in a “tee” configuration with tension-compression, torsion, and bending capabilities.

The execution of this command is convenient because the commands of Piping Models module enable to insert the models of selected pipe elements (curved and mitered bends, valves, flanges, reducers, bellows, and branch connections) into the prepared pipeline model framework (composed of straight elements). User selects the node where this element is to be created and enters the required parameter values. Thus the module executes the insertion of new pipe elements and shortens the original finite elements to create space for the new elements.

Inserting branch connection by applying TEE command creates three new pipe elements. Two form run pipe and one is on branch pipe. TEE command has one required parameter and optional parameters which are automatically set equal to default value in case they are not entered.

The required parameter is, as well as in other “Piping Commands”, node number at which the branch-connection is created. The second parameter distinguishes whether the branch connection is of UFT type or WT type. The default value is WT which would be incorrect for majority of the pipelines in nuclear power plants. Therefore, this parameter was selected as required and its value is exported into the parameter of TEE command.

The next two parameters of TEE command are ELEM and EINC. ELEM is an element number which will be given to the first of the new created elements and EINC is an increment of numbering of created elements. Their default values are a number greater by one than the current highest element number for ELEM and one for EINC. These values are suitable also for macro *tee.mac*, therefore they are not entered.

The following three parameters L1, L2, and L3 specify the length of the inserted elements modeling the branch connection. In macro *tee.mac* it is determined which of them come up to the run pipe elements and which is the length of branch pipe. For the run pipe elements chosen lengths are equal to two diameters of run pipe which is in accordance with the default value in TEE command. The branch pipe value is equal to one half of that diameter.

Length of branch pipe section is important because the entered value equal to one half of the run pipe diameter causes that the new node created at the branch pipe is located on the surface of the run pipe. Thus the second requirement of the code is satisfied. Then the properties of the new branch element must be modified. Since it must be rigid, it was chosen $E = 2.0 \cdot 10^{20}$ and $\rho = 0$. Thus it is achieved that the model becomes even more realistic than is required by the code because any other value of ρ than zero would bring additional mass compared to the mass of a real pipeline.

The third requirement holds only for UFT branches. We have solved it by creating new element of negligible length to the new node on the surface of the run pipe. The length of this element was chosen 0.1 times the diameter of the branch pipe d . For development of the new element, BELLOWS command from Piping Models module was applied. If no other parameter is entered, only an element of PIPE16 type of the desired length with the properties identical to the modeled pipeline is inserted to the selected node.

The purpose of application of BELLOWS command is to insert a new element with shortening of the adjacent elements. In the next step, keypoints which are essential for defining an element of BEAM4 type are created at the nodes of the new element and at the ending node of one of the run pipe elements of the branch connection. The inserted pipe element is deleted and instead of it, one BEAM4 element of rectangular cross-section is inserted by standard ANSYS procedure. Its cross-sectional moments of inertia are:

$$I_b = I_y / k_{x_3} \quad (4)$$

and

$$I_h = I_y / k_{z_3} \quad (5)$$

where I_y is cross-sectional moment of inertia in bending of pipe branch (if it was modeled by PIPE16):

$$I_y = \pi(D^4 - d^4) / 64 \quad (6)$$

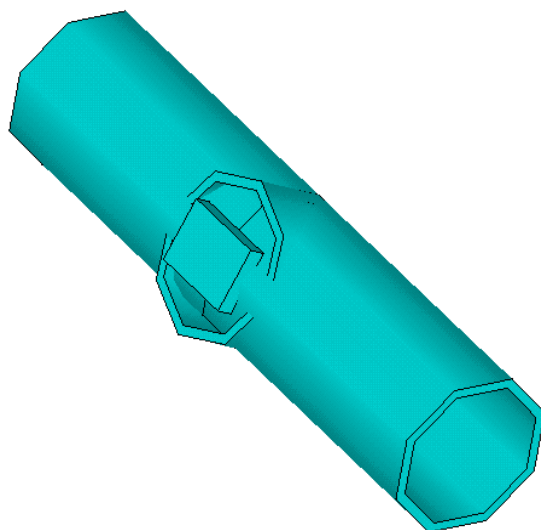


Figure 2 Model of the branch connection created by using macro *tee.mac*

Correct position of the rectangular cross-section is defined by third point which is located in the plane determined by the run pipe and branch pipe centerlines and is created using the third keypoint.

The computational model of the branch pipe developed by this procedure fully satisfies the requirements of the code (ASME Boiler & Pressure Vessel Code, 1992). Macro *tee.mac* will create it at any part of a pipeline where a branch pipe is modeled. The purpose of the use of BEAM4 element is to model the stiffness reduction of branch pipe due to effect of its attachment (weld) to thin wall of run pipe.

Example of the branch connection created by using macro *tee.mac* is presented in Figure 2.

4. Description of the developed macros

Macro *tee.mac*, which realizes the requirements of the code (ASME Boiler & Pressure Vessel Code, 1992), has 6 parameters:

ARG1 – number of the shared node of the run pipe and the branch pipe,

ARG2 – `UFT` or `WT` depending on the type of the inserted branch connection (if the parameter is entered incorrectly, error message is written out and the computation is terminated,

ARG3 – number of the branch pipe element which is connected to the run pipe,

ARG4 – number of the run pipe element at whose ending node the branch pipe is connected,

ARG5 – T_b' – thickness of the branch pipe according to fig. NB-3643.3(a)-1, ASME BPVC code (ASME Boiler & Pressure Vessel Code, 1992),

ARG6 – t_n – thickness of the branch pipe with included reinforcement according to paragraph NB-3686.5, ASME BPVC code (ASME Boiler & Pressure Vessel Code, 1992).

Parameters ARG5 and ARG6 are applied only for ARG2 = "UFT" while for ARG2 = "WT" they are not required. For a standard unreinforced branch pipe of "UFT" the values of these parameters equal to branch pipe thickness.

Macro *beg.mac* serves for proper defining of all input parameters for the computations, including material and section properties of the main part of the pipeline or the part of the pipeline which is modeled first. The macro must be modified before it is executed to correspond to the real modeled piping. It serves mainly as an instruction for setting the ANSYS program before beginning of the modeling of piping geometry.

After finishing macro *beg.mac* user can build the model of the entire piping system using ANSYS Piping Models module interactively in a common way. Only macro *tee.mac* with six required parameters must be applied instead of standard procedure TEE. The first two parameters are identical with those for procedure TEE and determine the node at which the branch connection is located and its type (UFT or WT) respectively. The next two parameters determine the element numbers of branch and one of run pipes. The last two parameters are thicknesses T_b' and t_n , that are needed for the determination of the flexibility of run pipe surface according to the code (ASME Boiler & Pressure Vessel Code, 1992).

Macro *tee.mac* will create file *tee.txt* in which are stepwise written data of all branch connections developed by using this macro and are necessary for the subsequent evaluation of the pipeline.

After the completion of the model of the pipeline and performing the static analysis for the defined loadings macro *rst.mac* is executed. It saves the text array which contains the data of the model geometry and the results of the performed static analysis.

Similarly macro *rdy.mac* saves the model geometry and the results of the performed dynamic analysis and macro *rse.mac* contains the model geometry and the results of the seismic analysis.

5. Modeling

Modeling of piping systems with the application of *tee.mac* is similar to modeling using Piping Models module.

First, it is necessary to develop either the entire framework of the modeled system or only its part. By applying commands BEND, MITER, VALVE, REDUCE, FLANGE, and BELLOW the desired pipe elements are developed at the selected nodes. Only macro *tee.mac* with appropriate parameters is executed instead of applying of TEE command. The commands and the macro can be executed in arbitrary order. It is possible to model the pipeline framework part by part while inserting the selected elements. Besides the mentioned commands it is also possible to use PSPRNG a PGAP commands which create flexible piping supports and flexible supports with gap.

Then stiff supports are added and loadings corresponding to type of analysis are defined. Piping Models module also allows to define internal and external pressure on the pipeline, temperature distribution, etc.

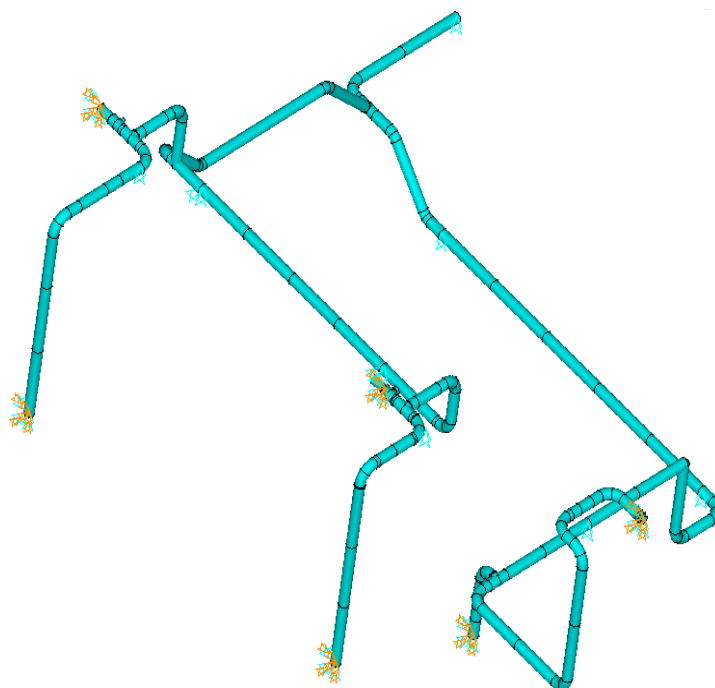


Figure 3 Model of the pipeline chosen for the demonstration of the application of the developed procedures

6. Computation analysis

After the completion of the piping model as it is described above, static or dynamic analysis is performed depending on the problem specification. For seismic response analyses, macro *rsm.mac* has been developed.

Figure 3 shows the pipeline model which was chosen to demonstrate the application of the developed modeling procedure and performing of the computations with the set up of the data files for further evaluations in Mathcad program.

7. Data export

For the purpose of the following piping evaluation, necessary data representing the model developed in ANSYS are saved in these files:

elem.txt – main table containing element properties

rc.txt – real constants specifying the geometric properties of the pipe elements of the model

node.txt – node coordinates

ko4.txt – type of pipe element defined by ANSYS variable Keyopt 4

tee.txt – number of branch connections and their properties

stat_mom.txt – calculated nodal bending and torsional moments from static analysis

dyn_mom.txt – calculated nodal bending and torsional moments from dynamic analysis

seis_mom.txt – calculated nodal bending and torsional moments from seismic analysis

8. Evaluation

The data of the model developed in ANSYS, which are exported to the data files listed above, are then loaded in the program set up in Mathcad. Based on the loaded data, individual pipeline elements are identified and resultant nodal stresses corresponding to the computed moments and the specified loading are determined. These obtained stress values are then compared with the allowable values determined in accordance with ASME BPVC (ASME Boiler & Pressure Vessel Code, 1992) code. If seismic adequacy of a pipeline is the concern, the aim of the evaluation is to determine the parameter called strength factor. Its value means how many times the current seismic loading can be multiplied until the allowable stress of the component is reached.

9. Calculation demonstrations

For a demonstration of the static analysis we chose the pipeline which is shown in fig. 3. This pipeline was loaded by horizontal force of magnitude $F_x = -300$ N and gravitational acceleration $g_z = 9.81$ m.s⁻². The applied loadings are displayed in fig. 4.

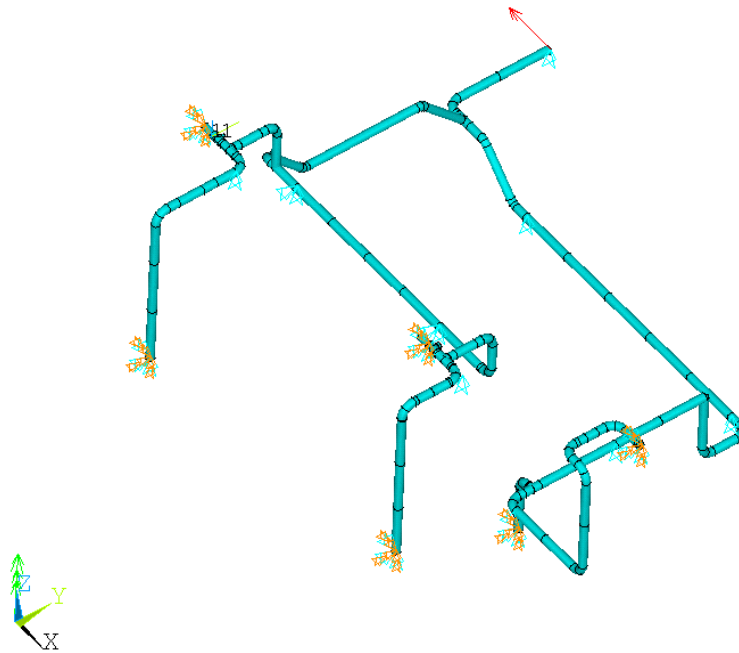


Fig. 4 Definition of the loadings for the static analysis

The resultant displacements for such static analysis loadcase specification are shown in fig. 5. One example of the mode shapes calculated by modal analysis is displayed in fig. 6.

Fig. 8 presents the maximum displacements of the pipeline subjected to the seismic loading specified by the response spectra shown in fig. 7. This computation was performed for the value of damping ratio equal to 0.3 which is applied identical for all 3 directions of acceleration

spectra. All 18 natural frequencies which were found within the range of 0-33 Hz were considered in the subsequent seismic response analysis.

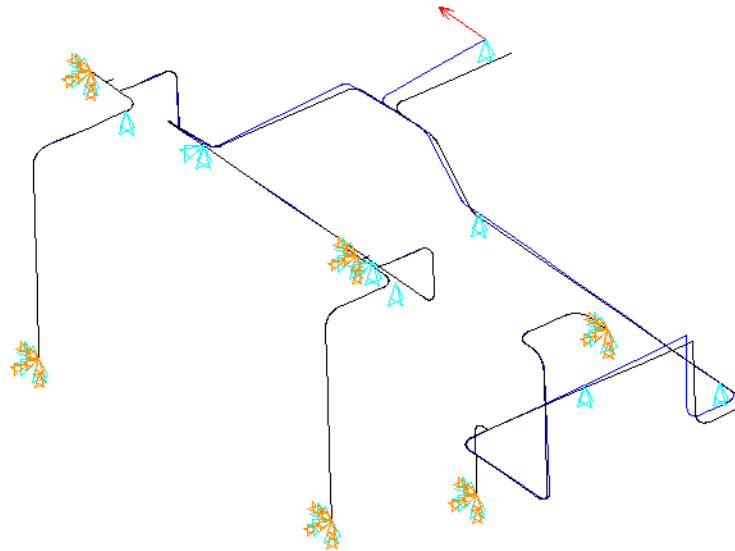


Fig. 5 Resultant displacements obtained by static analysis

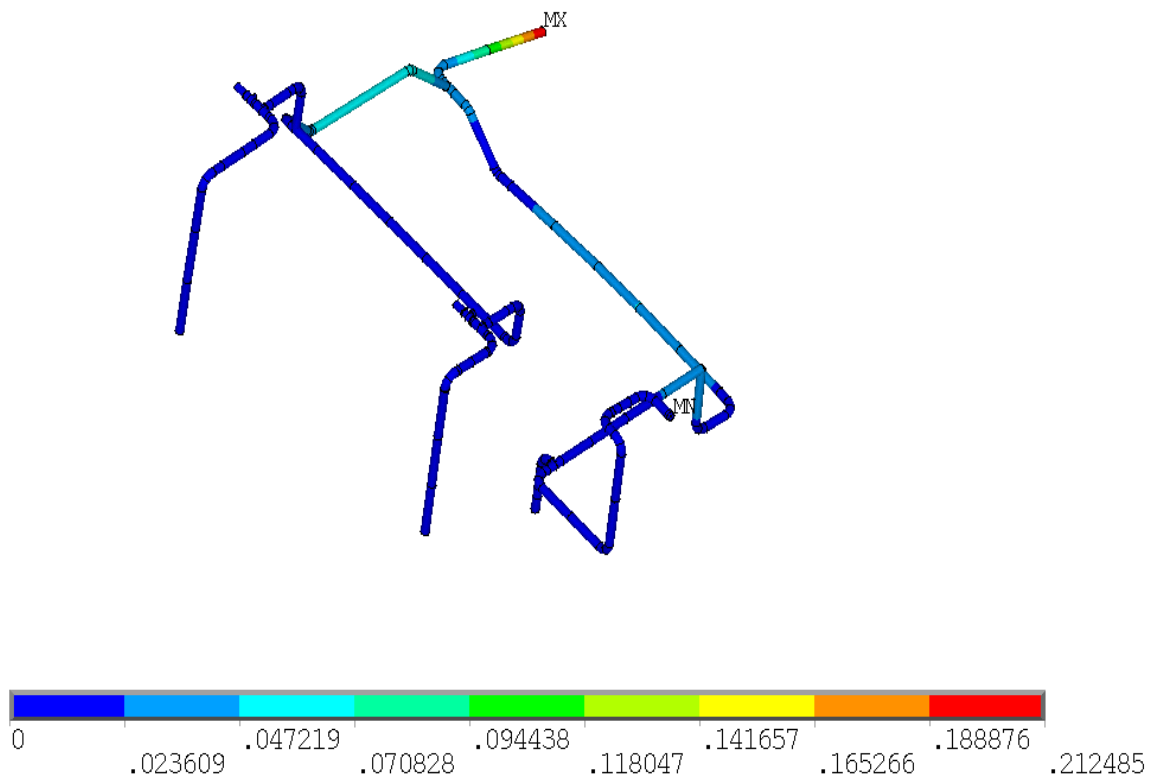


Fig. 6 Mode shape No. 5 - 12.4 Hz

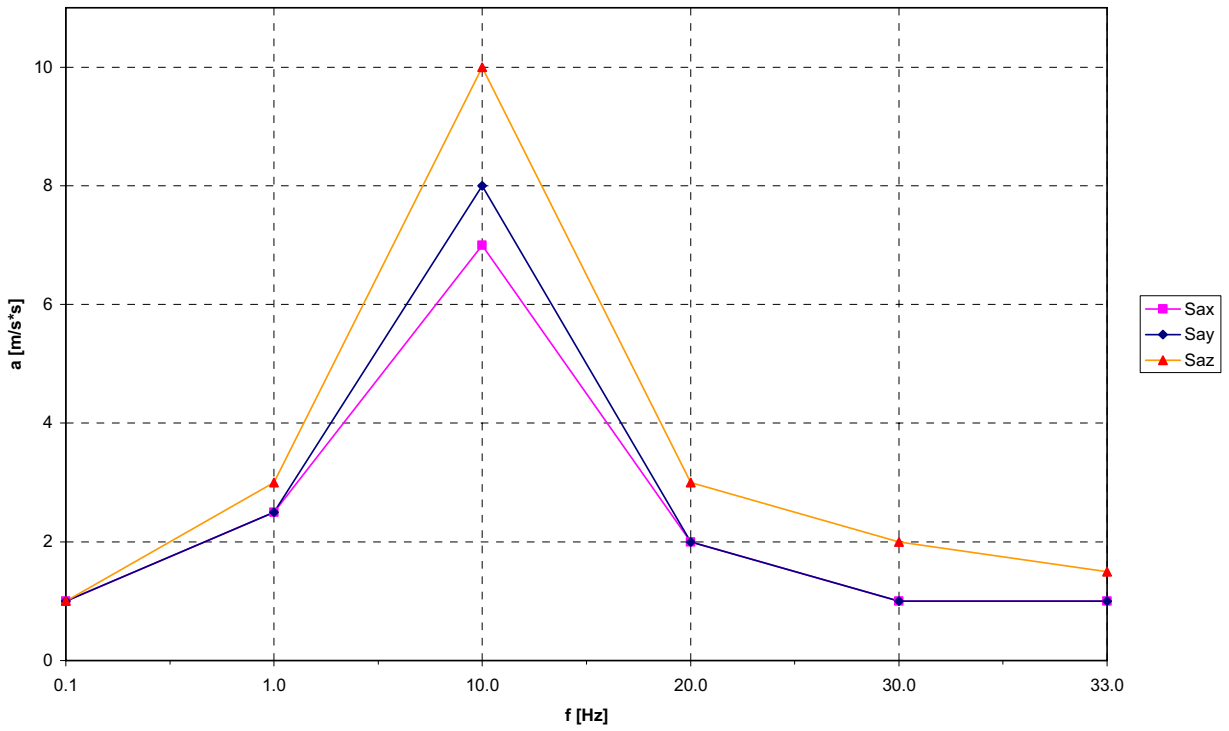


Fig. 7 Applied acceleration spectra for seismic response analysis

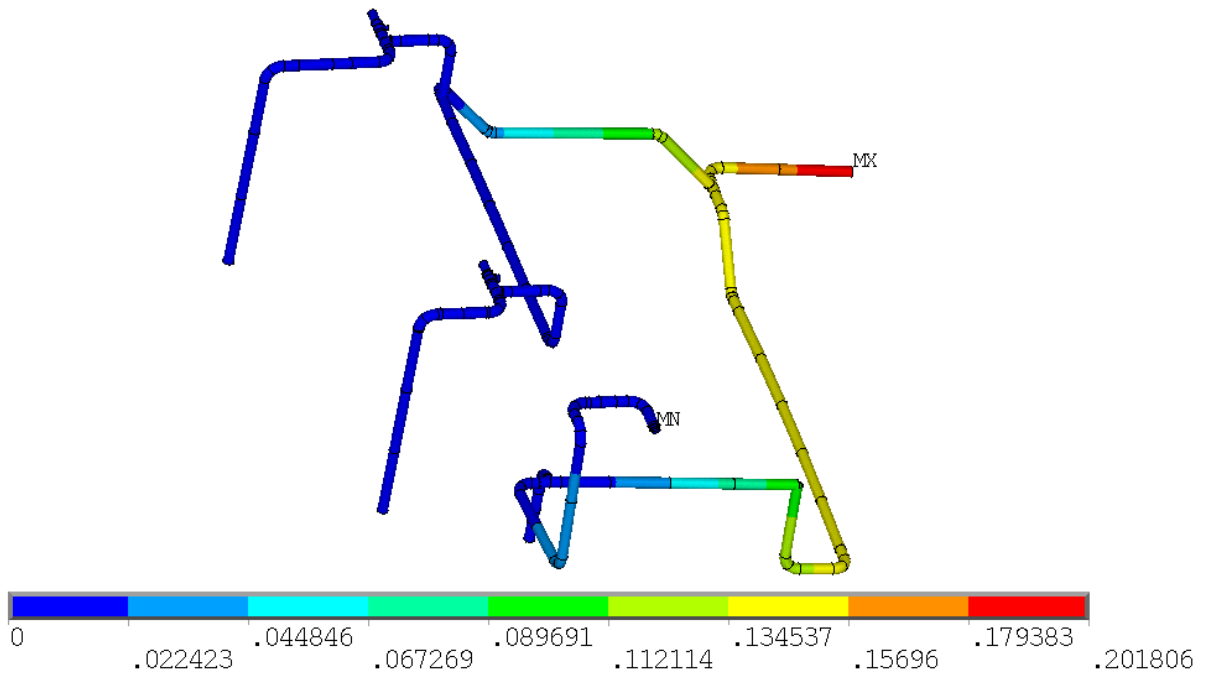


Fig. 8 Maximum displacements during the seismic loading specified by the chosen response spectra

10. Conclusion

This paper deals with the improvement of the first part of the piping evaluation procedure which is the development of the computational model including modeling of branch connections in accordance with ASME Boiler & Pressure Vessel Code, 1992. Detailed description of the following evaluation of static and seismic analyses is planned for Computational Mechanics 2009 conference.

11. References

- ASME Boiler & Pressure Vessel Code, (1992), Section III, Division 1 – Subsection NB, Class 1 Components. New York, USA.
- Stress Intensification Factors and Flexibility Factors for Unreinforced Branch Connections, (1998), EPRI Report No. TR-110996, Palo Alto, USA.
- Smith, Paul R., Van Laan, Thomas J. (1987) Piping and Pipe Support Systems: Design and Engineering. McGraw-Hill.
- Gupta, A. K. (1990) Response Spectrum Method in Seismic Analysis and Design of Structures, Blackwell scientific Publications.
- ANSYS, (2006), Ver. 10.0. „User Manual.“ ANSYS Inc. Canonsburg, USA.