

EFFECT OF MESH QUALITY IN FINITE ELEMENT ANALYSIS OF CRACK-TIP STRESSES IN A CIRCUMFERENTIAL SURFACE CRACK OF A PIPE ELBOW WELDMENT (*)

Robert B. Rainsberger¹

XYZ Scientific Applications, Inc., 2255 Morello Ave., Suite 220, Pleasant Hill, CA 94523, U.S.A.

Jeffrey T. Fong

U. S. National Institute of Standards & Technology, Gaithersburg, MD 20899-8910, U.S.A.

Pedro V. Marcal

MFACT Corp., 5297 Oak Bend Lane, Suite 105, Oak Park, CA 91377, U.S.A.

Abstract: Errors and uncertainty in finite element method (FEM)-based simulations originate from numerous sources. In this paper, we will address the errors and uncertainties due to two sources, namely, the mesh density, and the mesh quality. Our approach, using a super-parametric method, is to design a family of meshes of increasing degrees of freedom for a specific element type and a number of mean aspect ratios representing different mesh quality, and solve them in order to estimate the most probably accurate solution and its uncertainty of the FEM-based simulations. To obtain an estimate of the “correct” solution at “almost” infinite degrees of freedom, we introduce a nonlinear least squares fit algorithm based on a 4-parameter logistic distribution and apply to a sequence of at least five candidate solutions for a specific platform, element type, and mean aspect ratio.

1. Introduction

Errors and uncertainties in finite element method (FEM) based simulations are known to originate from at least eight sources: (1) The FEM computer *code or “platform”* using a specific numerical algorithm of approximation for solving a system of partial differential equations with initial and boundary conditions; (2) the choice and design of the FEM mesh using a specific grid spacing (or, *mesh density*), (3) same for a specific *element type*; (4) the “quality” measure of the FEM mesh such as the *mean aspect ratio*; (5) the uncertainties in the geometric parameters of a specific model; (6) the uncertainties in the physical and material property parameters of that model; (7) the uncertainties in the loading and constraint parameters, and finally (8) the uncertainty in the choice of the governing equations of that model.

In this paper, we will address the errors and uncertainties due to two of the above-listed eight sources, namely, (2) the *mesh density*, and (4) the *mesh quality*. Based on our earlier work [1, 2, 3], we use a super-parametric method to design a family of meshes of increasing degrees of freedom for a specific element type and a number of mean aspect ratios representing different mesh quality, and solve them in order to estimate the most probably accurate solution and its uncertainty of the FEM-based simulations.

To obtain an estimate of the “correct” solution at “almost” infinite degrees of freedom, we introduce a nonlinear least squares fit algorithm based on a 4-parameter logistic distribution and apply to a sequence of at least five candidate solutions for a specific platform, element type, and mean aspect ratio. The predicted “correct” solutions with uncertainties at **one billion degrees of freedom** are then compared with one another to yield a ranking of the solutions from the “most” to the “least” accurate, based on the “uncertainty metric.”

To illustrate this new approach, we present two examples, of which the first one has known solutions:

(*) Contribution of the U. S. National Institute of Standards & Technology. Not subject to copyright.
1 Corresponding author, E-mail address: r.rainsberger@yhoo.com (Robert B. Rainsberger)

(1) the first bending resonance frequency of a simple, isotropic, elastic cantilever beam, assuming a fixed platform and element type; (2) the elastic deformation of a pipe elbow with a circumferential surface crack at one of its two welds. The significance and limitations of this new approach as a tool for verification and validation of the finite element method will be discussed.

2. First Example – Cantilever Beam

Our first example is a cantilever beam used to demonstrate, in detail, how the mesh is varied to produce 5 candidate solutions that are then fit to a 4-parameter logistic function. The cantilever beam has a rectangular cross section and has a maximum tensile bending stress at the fixed end equal to 1500 MPa. We use an isotropic elastic material model. The length of the beam is such that simple beam theory applies. The ABAQUS finite element code is used to determine the fundamental frequency. The exact solution is 179.03kHz. Each mesh in the sequence has an increased number of Degrees Of Freedom (DOF) while the Mean Aspect Ratio (MAR) is held at a constant. In this detailed example, MAR is at 6.63. We start with a mesh with 1535 DOF in Figure 1. ABAQUS determined the fundamental frequency to be 182.045 kHz.

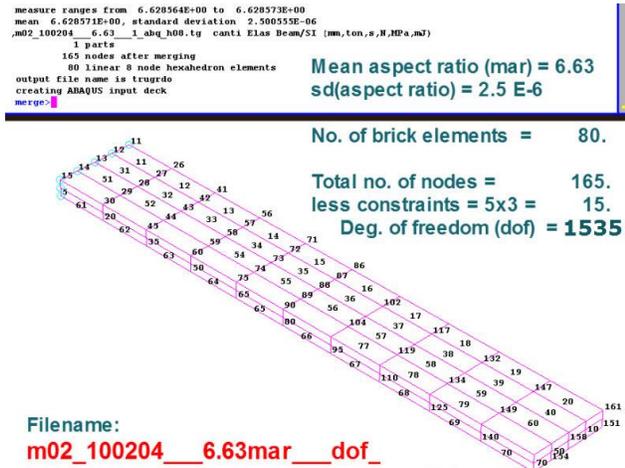


Figure 1: 1535 DOF

In Figures 2, the mesh density is increased to the maximum for this example with 36,543 DOF. ABAQUS predicted a fundamental frequency of 181.077 kHz.

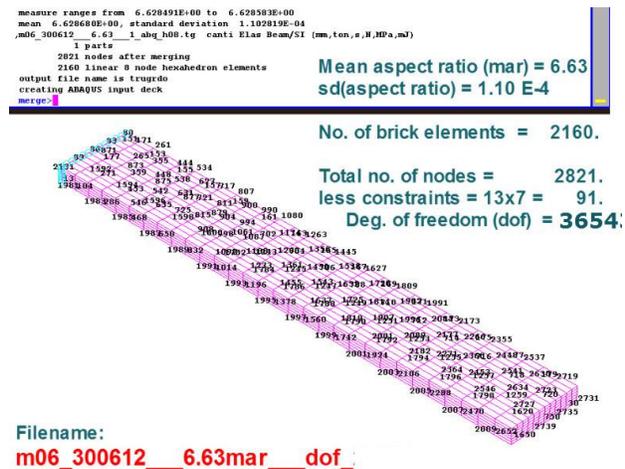


Figure 2: 36.543 DOF

Five versions of the mesh are used, each with a larger number of DOF than the previous. These five 2D data points (DOF vs. calculated Fundamental Frequency) are plotted on a logarithmic scale.

Table 1 lists the DOF and corresponding predicted fundamental frequency (in kHz) for each mesh.

Table 1

1535	182.045
4854	181.682
11155	181.402
21398	181.211
36543	181.077

Then a non-linear least squares calculation determines the four parameters in a logistic function which is then used to extrapolate to the point of convergence. The results of the ABAQUS hexa 8 test is shown in Figure 3.

A variation of the mesh was produced with a MAR of 5.52 and 5 mesh densities were chosen with 1,833, 5,808, 13,359, 25,638, and 43,797 DOF, respectively. The same calculations were performed on these 5 meshes using ABAQUS. The results are shown in Figure 4. As one might expect, the predicted fundamental frequency is slightly closer to the exact solution of 179.03.

This experiment was repeated 5 more times using MAR of 4.14, 3.68, 3.31, 2.76, and 1.16. Figures 5-9 show the results of these test.

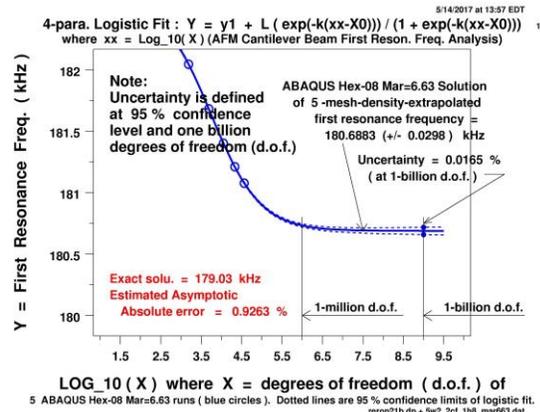


Figure 3: Four Parameter Logistic Fit

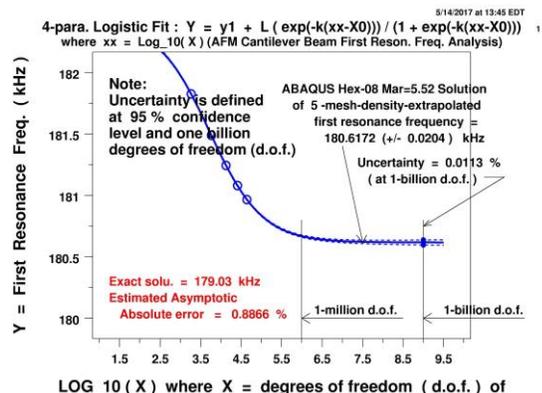


Figure 4: Logistic Function for MAR=5.52

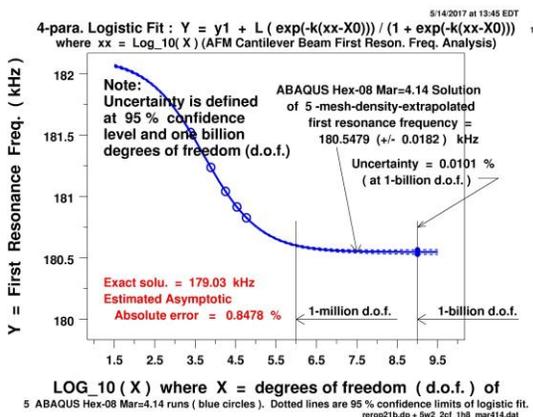


Figure 5: Logistic Function for MAR=4.14

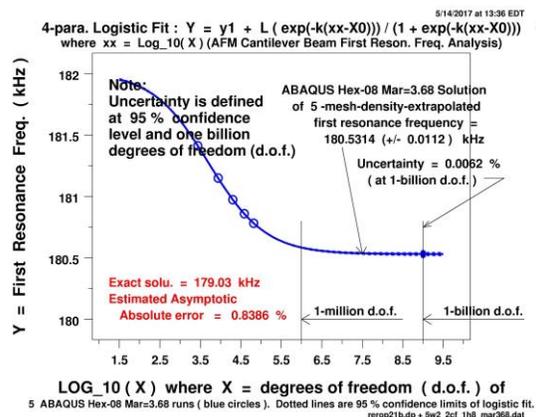


Figure 6: Logistic Function for MAR=3.68

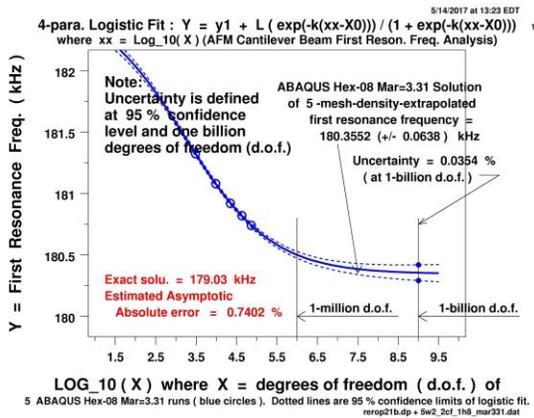


Figure 7: Logistic Function for MAR=3.31

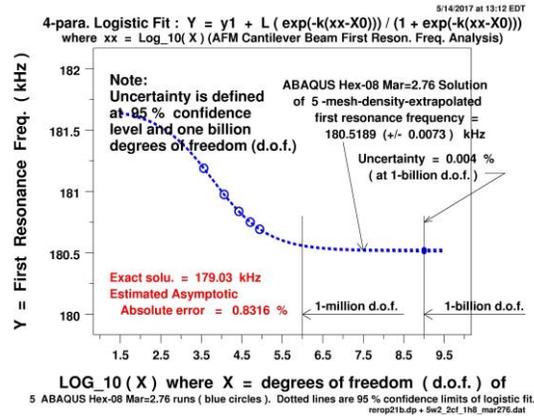


Figure 8: Logistic Function for MAR=2.76

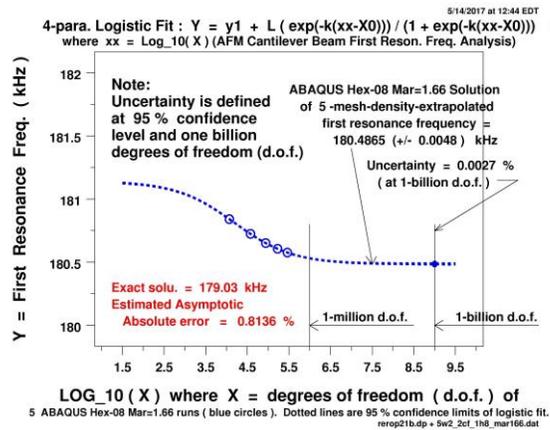


Figure 9: Logistic Function for MAR=1.66

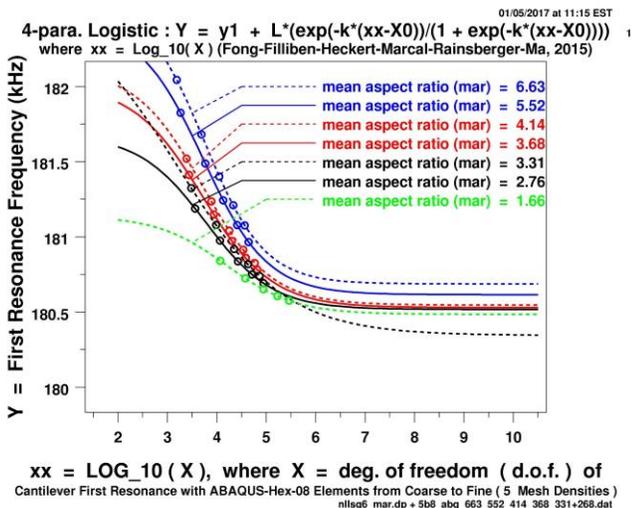


Figure 10: 7 Nonlinear Fit of FEM Solution of First Resonance Frequency of a Cantilever Beam

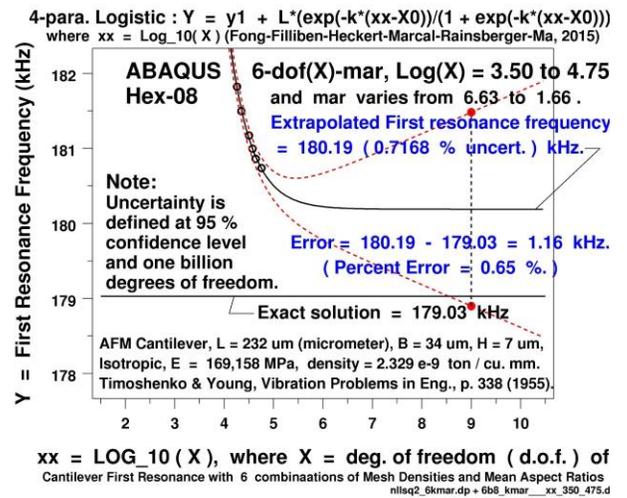


Figure 11: FEM Solution with Uncertainty for First Resonance Bending Frequency of a Cantilever Beam

3. Super-Parametric Mesh Generation

This first example demonstrates a method to determine the accuracy of a FEM. It is also clear that the quality of the mesh plays an important role in the accuracy of the simulation. Of course, the errors involved in the simulation due to mesh quality are small, but this should be expected for such a simple problem. But one thing is certain. There were 35 different hexa meshes built for this experiment. In addition, each family of 5 meshes all had the same MAR and a monotonically increasing DOF. It is a chore to carefully build that many meshes. The only way that the method will be followed is if the meshes can be generated with ease. We used the super-parametric mesh generator called **TrueGrid**[®] to generate all 35 meshes. After the first one was built, additional meshes only required the change of a single parameter and a rerun. **TrueGrid**[®] is super-parametric because, with a single command change, it will generate a mesh for almost any popular FEM simulation code. With a single command change it will generate all linear tetra (4 nodes), linear hexa (8 nodes), quadratic tetra (10 nodes), quadratic hexa (20 nodes), or triquadratic hexa (27 nodes).

The method described here can be used to compare the accuracy of the same element type across multiple FEM simulation codes, as is done in the second example in this paper. Or it can be used to determine the accuracy by comparing the different element types within a single FEM simulation code. Or one can do as we did in this first example to compare the effect of the mesh quality on the solution.

4. Second Example – Crack in a Pipe Elbow Weldment

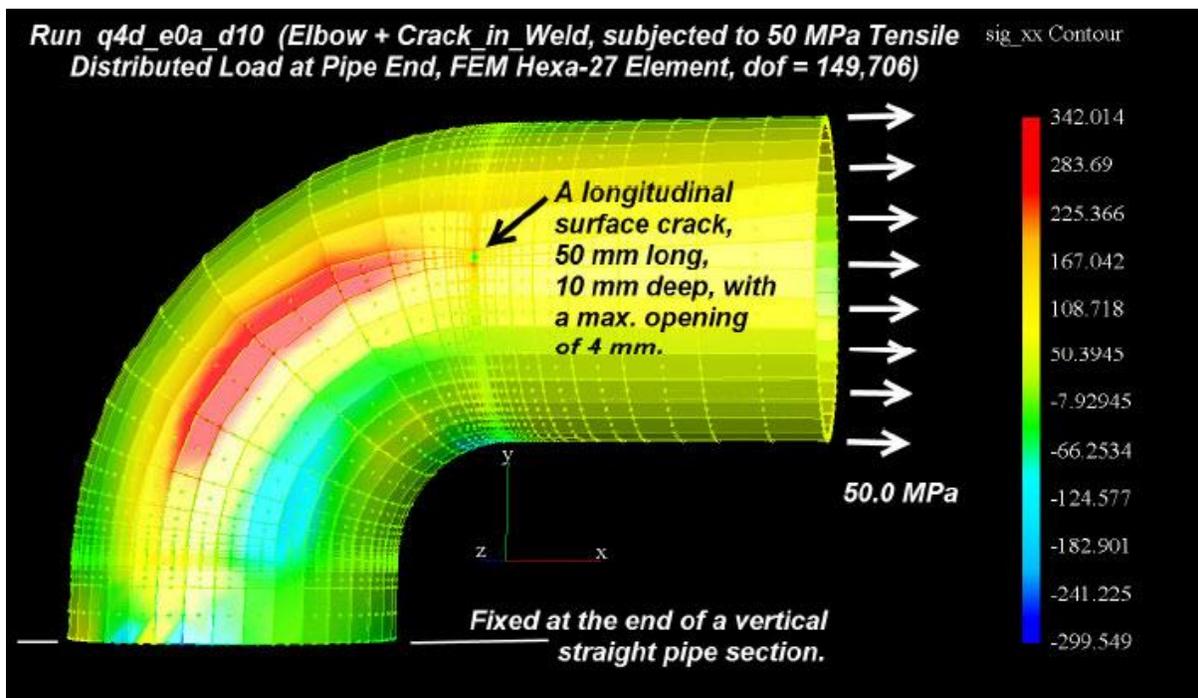
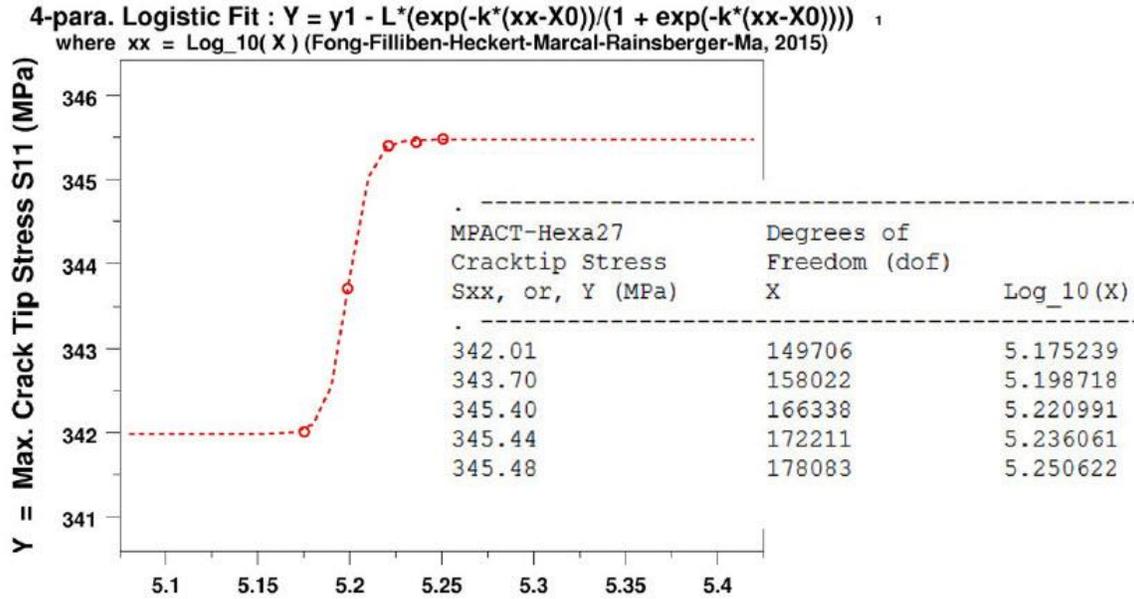


Figure 12: A Finite Element Solution for Stresses in a Cracked Pipe Elbow

Figure 12 shows the results of a FEM simulation using Mpact. This mesh of a pipe elbow weldment with a vertical surface crack along the weld is formed from 27-node hexa elements (triquadratic). There are 149,796 degrees of freedom. One end of the pipe is constrained with a force applied to the other end. This experiment is done 5 times, each time with a great number of DOFs then the previous time. As was done in the case with the cantilever beam, a logistic function is fitted to these 5 data points (see Figure 13).



$xx = \text{LOG}_{10}(X)$, where $X = \text{deg. of freedom (d.o.f.) of}$

Figure 13: Nonlinear Least Squares Fit of 5 MPACT-Hexa-27 Solutions

Notice that the gap between the lower and upper asymptotes of this graph (3.5 MPa) is relatively small. This indicates that a model using the MPACT Hexa 27 (triquadratic 27 node brick) elements converges quickly as the mesh density increases.

5. Pipe Elbow Weldment Mesh Generation

There are a small number of parameters that define the model of the Elbow Pipe Weldment with a surface crack. They are declared using the para command found in the session file for **TrueGrid**[®] as seen below:

```
c      CONTROL MESH DENSITY
para
  d 10          c Mesh density - an integer greater than 1
;

c      DEFINE THE GEOMETRY
para outtyp 4   c output type: 1 for ABAQUS, 2 for ANSYS,
                c 3 for LSDYNA, 4 for NASTRAN, 5 for MPACT
  r1 300        c large outer radius of the elbow
  r2 150        c outer cross section radius
  th 25         c thickness
  rlen 300      c length of straight pipe to the right
  dlen 100      c length of straight pipe going down
  wth 25        c width of the weld region (0 for no weld region)
  gap 3         c initial gap
;
```

```

c      SELECT OUTPUT PROPERTIES
para outtyp 4          c output type: 1 for ABAQUS, 2 for ANSYS,
                      c 3 for LSDYNA, 4 for NASTRAN, 5 for MPACT
                      elmtyp 3          c 1 for linear tets, 2 for quadratic tets,
                      c 3 for linear hex, 4 for quadratic hex
                      youngs 3.0e+07   c Young's modulus
                      poissons .3      c Poisson's Ratio
                      massdens 2.2     c mass density
;

c      MESH PARAMETERS
para
nqr [%d*3]            c number of elements in a quarter of a circle
                      c around the pipe - odd number
wcpn [1+%nqr/3]      c width in number of nodes of the
                      c coupon containing the gap - even number
                      c exceeding wgap
wgap [%wcpn-2]       c width in number of nodes that form the gap
                      c even number
nth 7                 c number of elements in thickness of the pipe
hgap 1                c height in number of nodes that form the gap
                      c smaller than nth
nrlen %nqr           c number of elements in the length of pipe to
                      c the right
nelbow %nqr          c number of elements along the half elbow
ndlen [%nqr/3]       c number of elements in the length of pipe
                      c going down
gang [90/%nqr]       c angle between radial mesh lines
;

```

The first parameter, d, is the density parameter. It is setup to be easy to use. When d is set to 3 (see Figures 14 and 15) we get the following coarse hexa 8 mesh which is similar to the mesh shown in Figure 12.

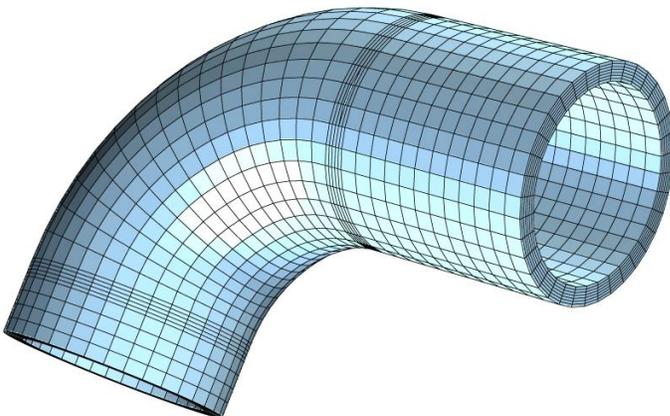


Figure 14: Elbow Pipe Coarse Mesh

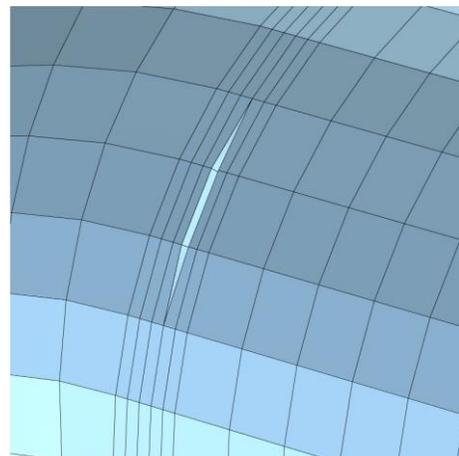


Figure 15: Elbow Pipe Crack Coarse Mesh

If *elmtyp* is set to 1, we would get the mesh in Figure 16. The same nodes are used in this tetra mesh than what were used in the hexa 8 mesh in Figure 14.

We use the same nodes because when making comparisons between different elements types, it is important not to introduce unnecessary variations in the two meshes.

These are just some of the options available with a super-parametric mesh generator. One of the most important options is the variation in the mesh density without changing the topology of the mesh.

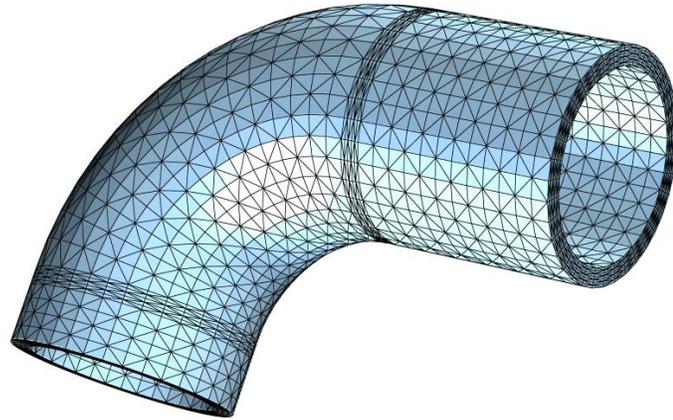


Figure 16: Equivalent Tetra Mesh of Elbow Mesh

In order to determine the effect of the mesh quality on the FEM solution, we will vary the Mean Aspect Ratio (MAR). We need to generate a family of meshes where the aspect ratio is monotonically decreasing. One way to do this is to increase the mesh density by changing the parameter *d*. We start with *d*=2. The resulting mesh is seen in Figures 17 and 18. There are two aspect ratios calculated. The first is aspect ratio of the elements nearest to the crack. The second aspect ratio is for the entire model. These meshes are designed so that the two MAR are always close. Table 2 shows all the MAR for the 11 meshes.

This method has the disadvantage that the MAR goes down as the mesh density goes up. An alternate method where the MAR goes down without a significant increase in mesh density will be demonstrated next.

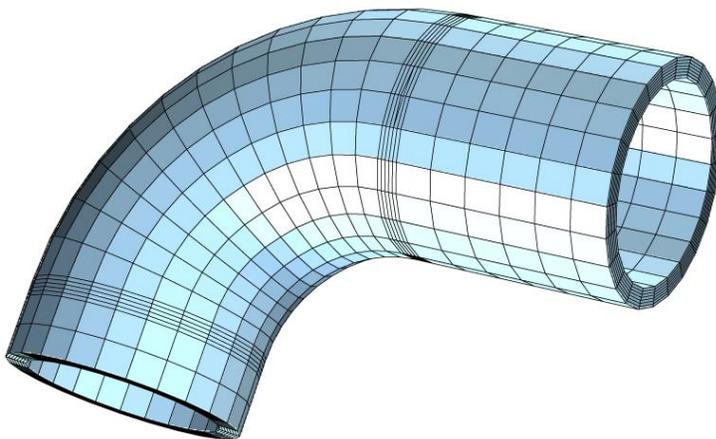


Figure 17: Elbow Pipe *d*=1

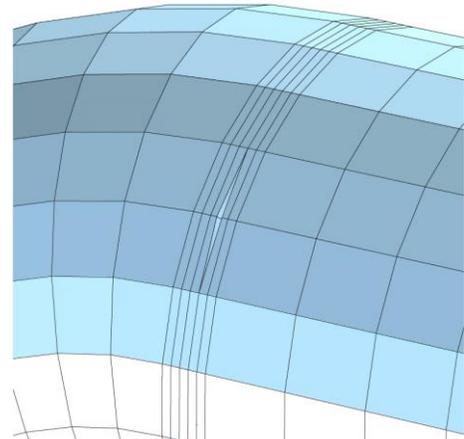


Figure 18: Crack *d*=1

Table 2

Density	Number of Nodes	Crack MAR	Full MAR
2	5,976	10.19	11.67
3	11,838	6.81	7.88
4	19,593	5.11	5.96
5	29,310	4.09	4.79
6	40,926	3.14	4.01
7	54,442	2.92	3.44
8	69,902	2.55	3.02
9	87,277	2.27	2.69
10	106,590	2.04	2.42
11	127,789	1.86	2.20
12	150,928	1.70	2.02

We now show the other extreme with 150,928 nodes. We set $d=12$. The resulting mesh is shown in Figures 19 and 20. This last mesh was included so that the MAR did not exceed 2. We chose 2 because it is recommended for several FEM codes as the maximum aspect ratio.

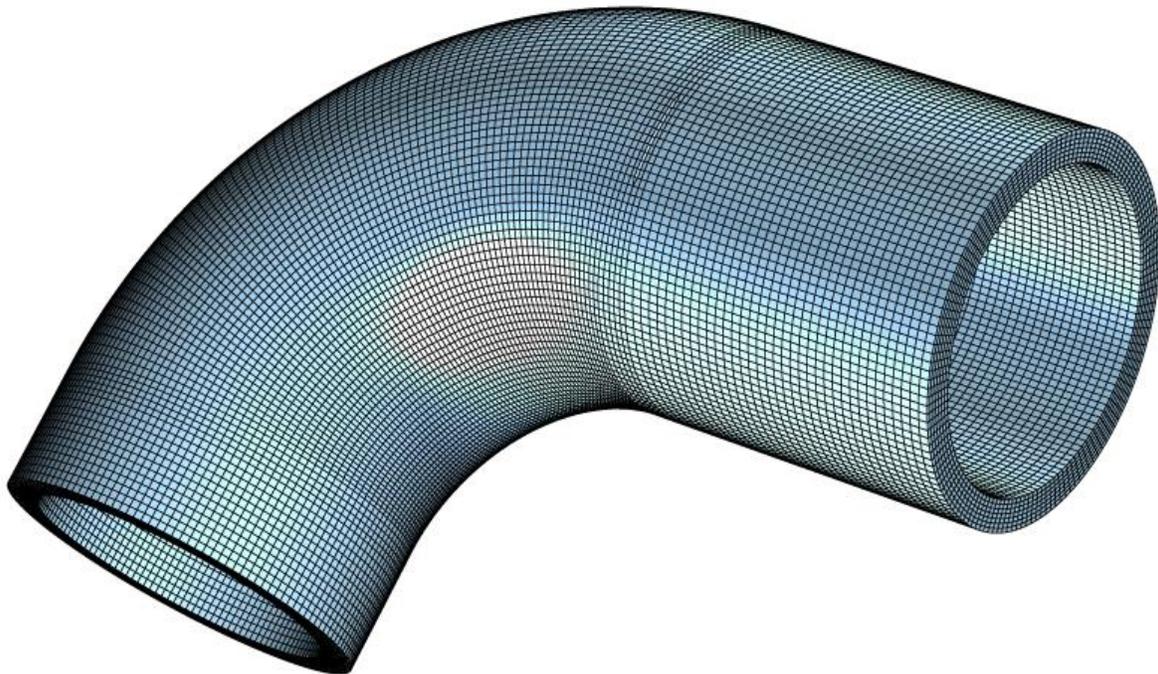


Figure 19: Elbow Pipe $d=12$

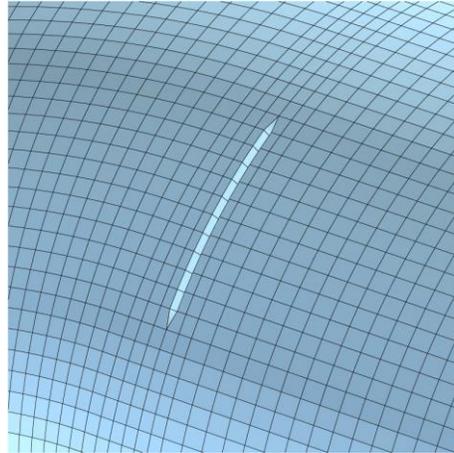


Figure 20: Crack d=12

An alternative to increasing the mesh density while decreasing the MAR is to create a transition region between the dense mesh of the region containing the crack and the coarse mesh elsewhere. We show a progression of regions in the mesh for such a topology in Figures 21 through 24.

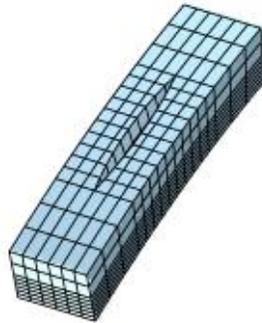


Figure 21: High Density Inner Region

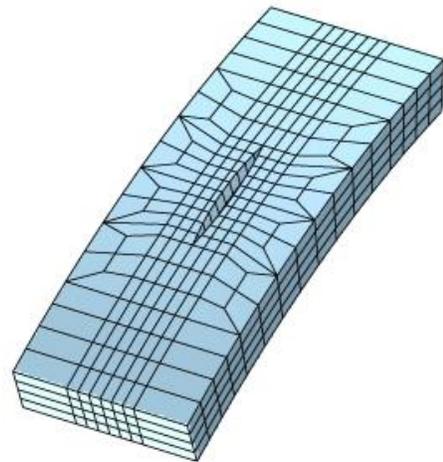


Figure 22: 1st Transition region Added

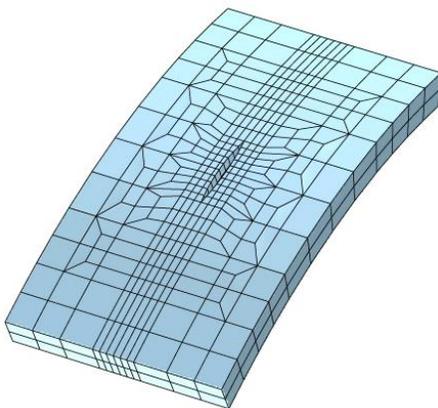


Figure 23: 2nd Transition Region Added

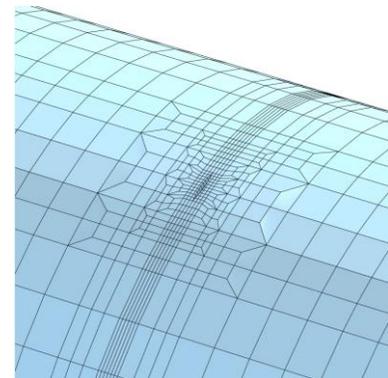


Figure 24: 3rd Transition Added

Finally, we have the example of the elbow pipe mesh (Figure 25) with crack and transitions with mesh density doubled. As before, the mesh density can be increased everywhere by changing one parameter. Alternatively, additional layers of transitions can be added to only increase the mesh density in the area of interest, in this case the crack, without increasing the overall mesh density. It is not known what effect the transition regions will have of the accuracy of this problem. It is the goal of this project to establish the correct solution using the first set of meshes and then to determine the effect of the transitions topology on the accuracy of the FEM solution.

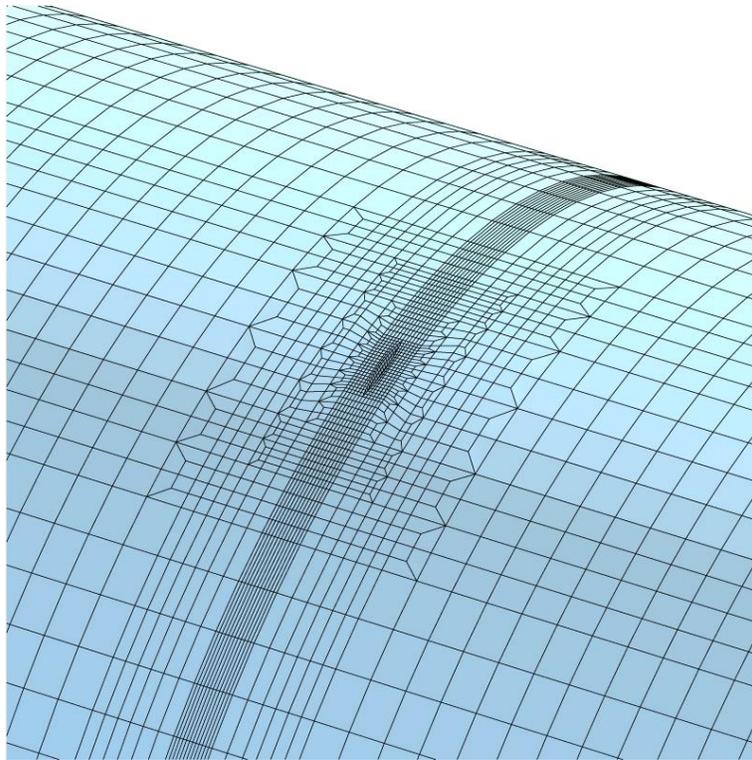


Figure 25: Crack in Elbow Pipe Doubled With Transitions

6. Conclusions

With a super-parametric method, one can generate a family of FEM meshes. When the FEM results are fit to a 4-parameter logistic function, the error in the simulation can be accurately estimated. In particular, the example presented demonstrates the effect of mesh density and quality on the accuracy of the FEM simulation.

Acknowledgments

We wish to thank James J. Filliben, N. Alan Heckert, and Li Ma, all of the U.S. National Institute of Standards and Technology, for technical assistance during the course of this investigation.

References

- [1] P.V. Marcal, J.T. Fong, R. Rainberger and L. Ma, Finite Element Analysis of a Pipe Elbow Weldment Creep-Fracture Problem Using an Extremely Accurate 27-Node Tri-Quadratic Shell and Solid

Element Formulation, Proc. 14th International Conf. on Pressure Vessels Technology, ICPVT-14, Sep. 23-26, 2015, Shanghai, China, 2015, Procedia Engineering 130 (2015) 1110-1120.

[2] J.T. Fong, J.J. Filliben, N.A. Heckert, P.V. Marcal, R. Rainberger and L. Ma, Uncertainty Quantification and Extrapolation of Stresses in a Cracked Pipe Elbow Weldment Using a Logistic Function Fit, a Nonlinear Least Square Algorithm, and a Super Parametric Method, Proc. 14th International Conf. on Pressure Vessels Technology, ICPVT-14, Sep. 23-26, 2015, Shanghai, China, 2015, Procedia Engineering 130 (2015) 135-149.

[3] R. Rainsberger, J.T. Fong and P.V. Marcal, A Super-Parametric Approach To Estimating Accuracy and Uncertainty of the Finite Element Method, Proc. ASME Pressure Vessels and Piping Conf., PVP2016-63890, July 17-21, 2016, Vancouver B.C., British Columbia, Canada.