PVEng

Pressure Vessel Engineering Ltd
120 Randall Drive, Suite B
Waterloo, Ontario, Canada
N2V 1C6

www.pveng.com
info@pveng.com
519-880-9808

Pressure Vessel Engineering Ltd. provides: ASME Vessel Code Calculations - Finite Element Analysis (FEA) - Solid Modeling / Drafting - Canadian Registration Number (CRN) Assistance

---

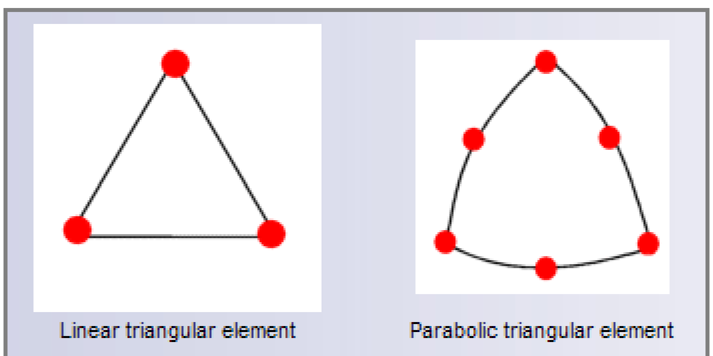
This is part of a series of articles that examines the ABSA (Alberta Boilers Safety Association) requirements on writing FEA reports. These guidelines can be found at: http://www.absa.ca/Forms/AB-520%20Finite%20Element%20Analysis%20(FEA)%20Requirements.pdf. The use of 2nd or higher order elements is one of the requirements.

Pressure Vessel Engineering uses CosmosWorks for Finite Element Analysis. It is expected that these results would also be applicable to other FEA programs.

Why use 2nd Order Integration Elements?

1) Because ABSA tells us to?
2) Because that is the default Cosmos Designer Setting?

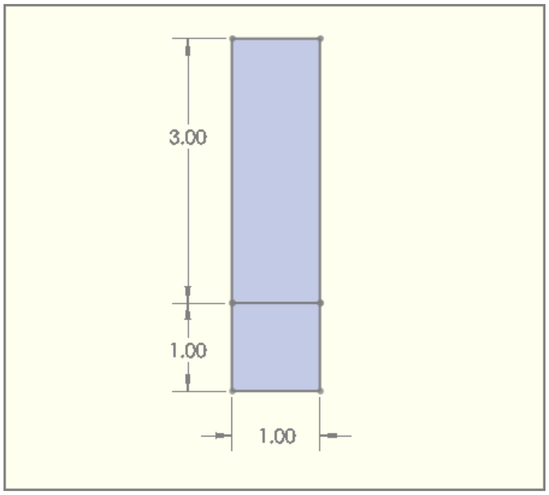
1st Order integration is found in the Mesh Options box under quality. The **Draft** option produces first order elements. **High** option produces 2nd order or higher – the default option. Integration beyond 2nd degree has to be chosen through the analysis properties window. 2nd order is the highest order available for shell elements.



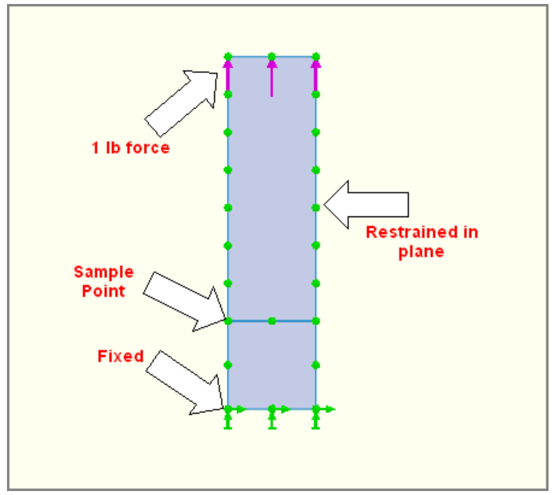
First and second order shell elements – 2nd order adds mid-side nodes

Problem 1 – Shell Elements in Tension

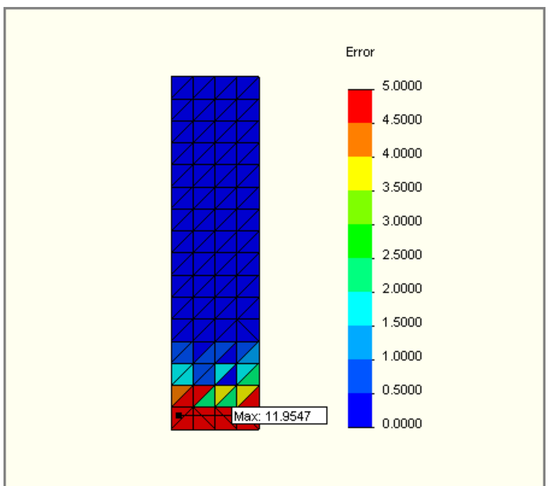
Which elements will produce better results for a simple tension load? The sample problem below is worked out in both 1st and 2nd order elements.



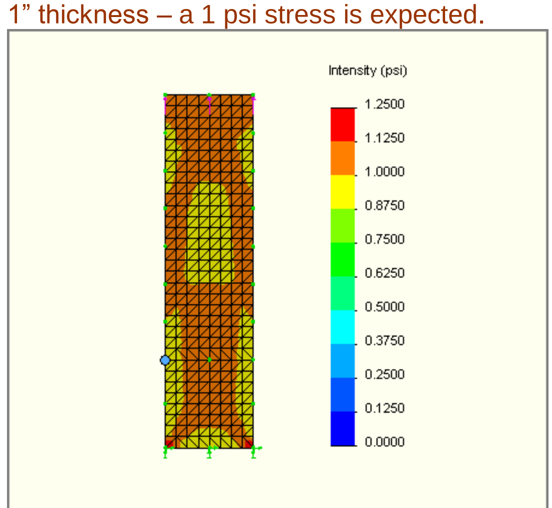
The model - a bar 1" wide x 4" long - it is split at 1" to make a sampling point.



The bottom is fixed and a 1lb tension load is applied to the top. The model is meshed at 1" thickness – a 1 psi stress is expected.



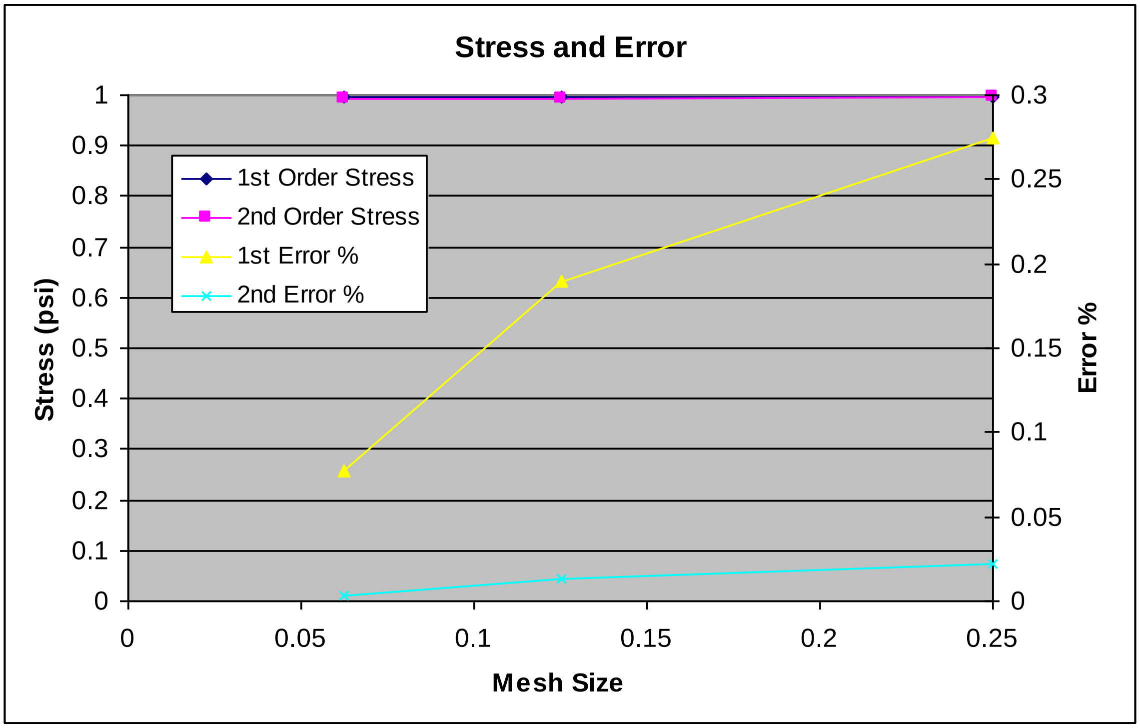
Typical error plot: ¼" mesh 1st order elements shown



Stress Intensity plot for 1/8" 2nd order elements.

Mesh	1st Order Stress	2nd Order Stress	1st Error %	2nd Error %
0.25	0.9946	0.9945	0.275	0.0225
0.125	0.9951	0.9942	0.19	0.0133
0.0625	0.9947	0.9942	0.0774	0.0031

Stress and Error Results

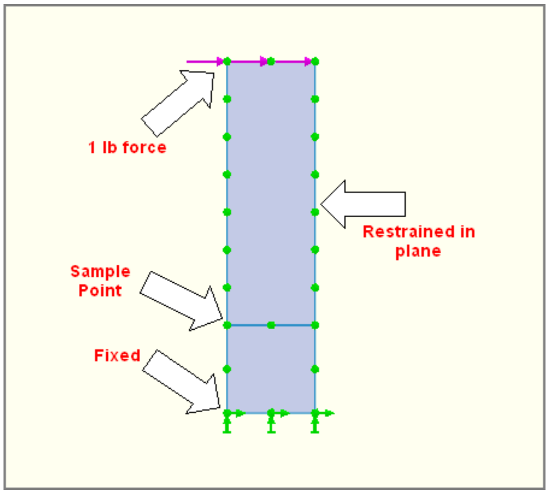


Graph of stress and error plot for the sample point. The stress values are practically identical; however, the 1st order elements have a much higher reported error level.

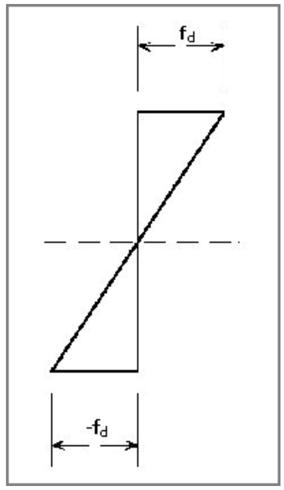
For this problem with a simple stress distribution, both the 1st and 2nd order elements produce excellent results as the mesh changed from ¼ to 1/16" size.

Problem 2 – Shell Elements in Bending

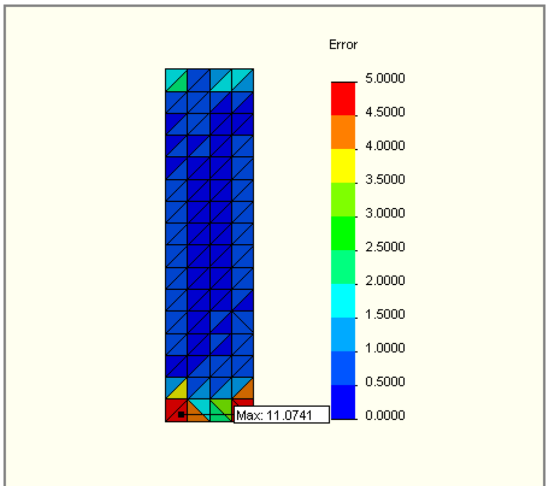
Using the same model from sample #1, the 1 lb tension load is changed to a 1 lb sideways or bending load. The moment of inertia is bh^3/12 = 1/12 in^4. The distance from neutral axis is 0.5". The moment at the sample point is 3 in*lbs . The expected stress at the sample point is Mc/I = 3*0.5/(1/12) = 18 psi.



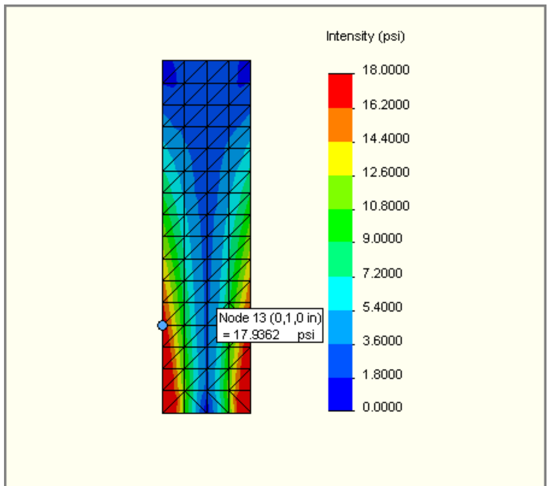
Same model – load is now horizontal to create a bending load



The expected stress distribution for a bending load



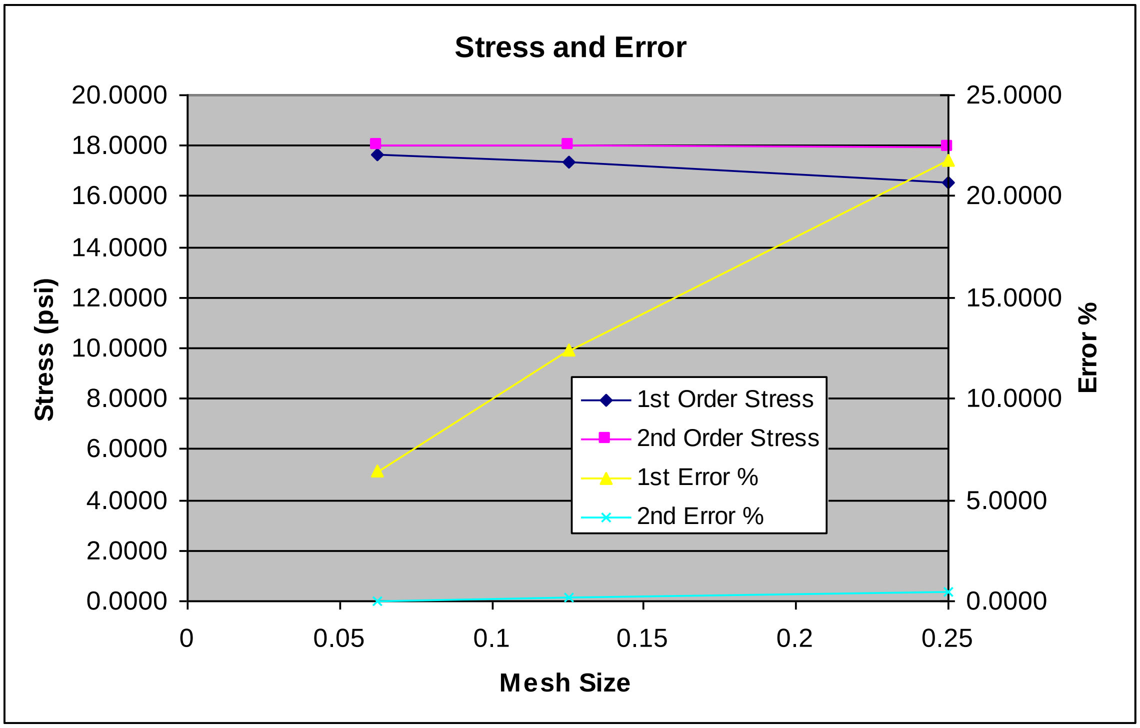
Error plot for ¼" 1st order elements.



Stress plot for ¼" 1st order elements.

Mesh	1st Order Stress	2nd Order Stress	1st Error %	2nd Error %
0.25	16.5131	17.9362	21.7930	0.5048
0.125	17.3167	17.9844	12.3823	0.1411
0.0625	17.6681	17.9953	6.4582	0.0329

Stress and Error Results



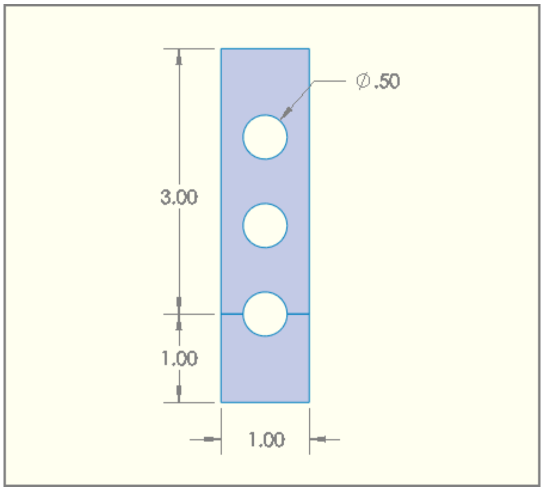
Graph of stress and error plot for the sample point. The First order elements have much higher real and reported error levels.

The stress pattern in this bar is a simple linear distribution – but the 1st order elements do a lousy job of representing it. The second order elements did a good job, even at the coarsest mesh size.

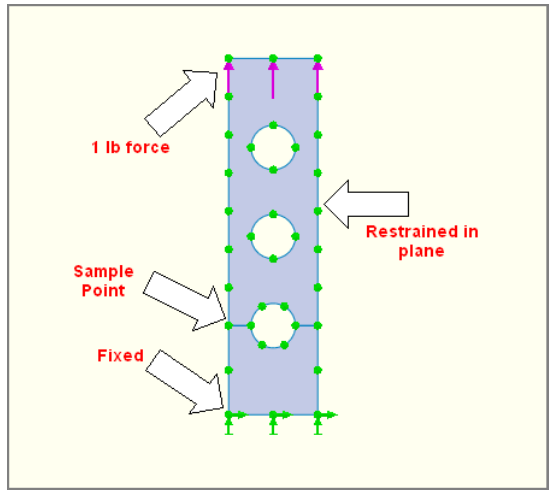
The reported error in all cases is much higher than the real error. For example the reported stress for the 1st degree elements at ¼" mesh is 16.5131 psi, theoretical stress is 18 psi. The real error is 8.3%, but it is reported at 21.8%. This over estimation is true for all the reported errors.

Problem 3 – Complex Stress in Shell Elements

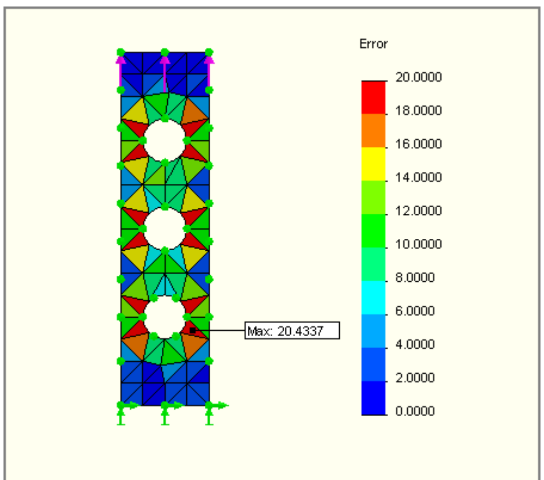
Simple uniform or linearly varying stresses do not often show up in real world FEA problems. How do the 1st and 2nd order elements handle more complex stress patterns?



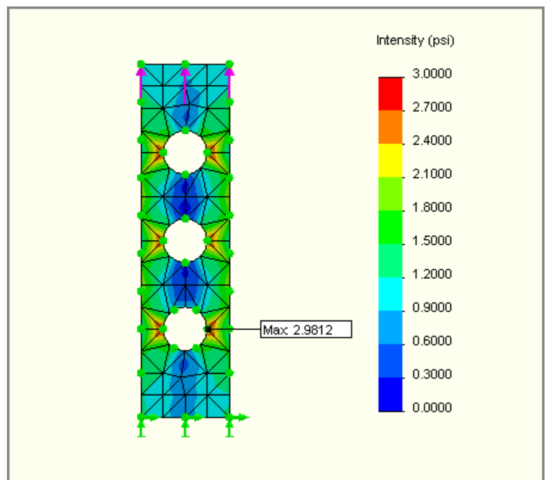
The model - a bar 1” wide x 4” long - it is split at 1” to make a sampling point.



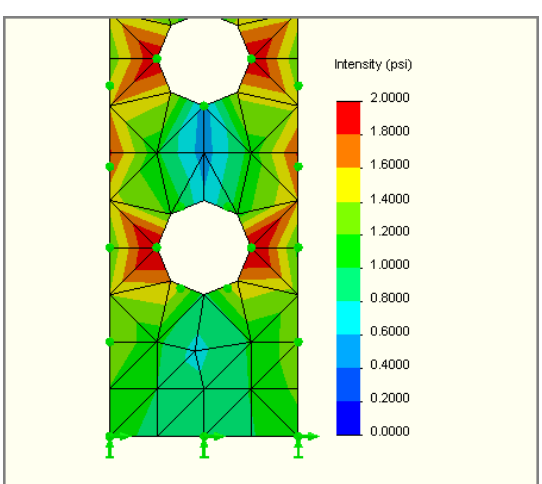
The bottom is fixed and a 1lb tension load is applied to the top. The model is meshed at 1” thickness – a complex stress pattern is expected.



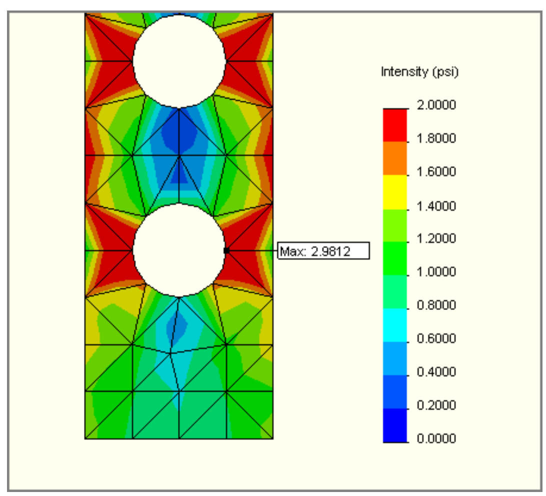
Error plot for ¼” 1st order elements.



Stress plot for ¼” 1st order elements.



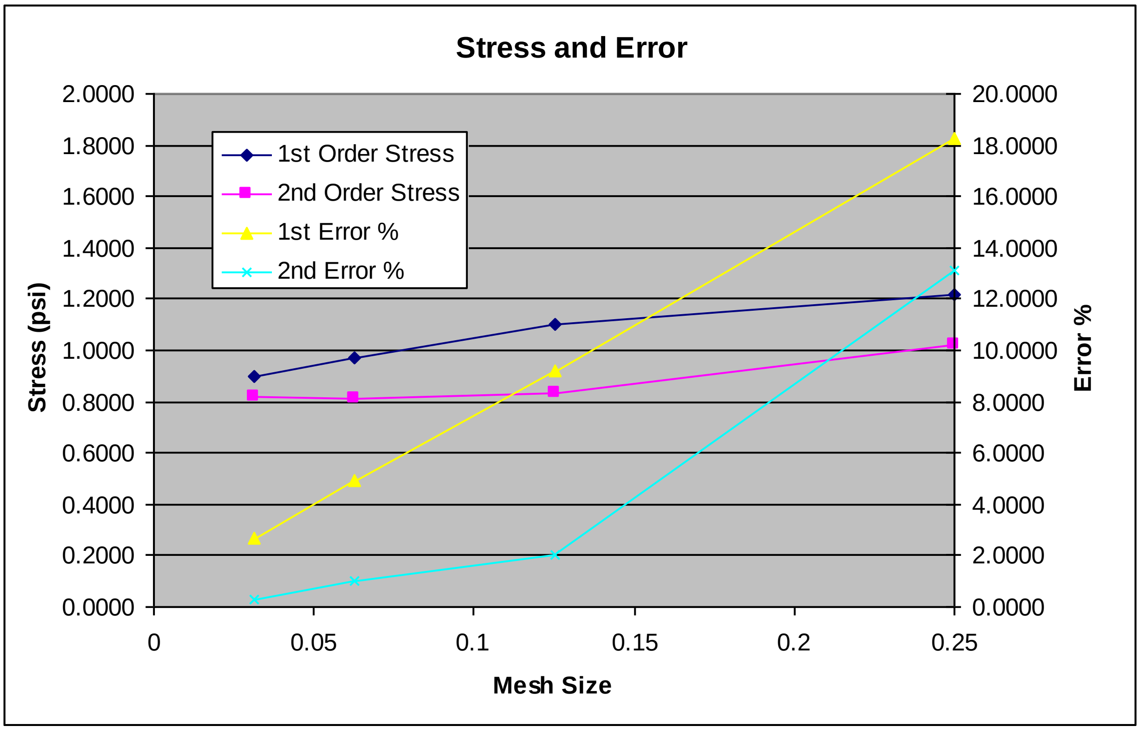
Mesh close-up – 1st order ¼” elements



Mesh close-up – 2^{nd} order ¼” elements –
note the better looking holes

Mesh	1st Order Stress	2nd Order Stress	1st Error %	2nd Error %	1st DOF	2nd DOF
0.25	1.2191	1.0210	18.2405	13.1510	385	1305
0.125	1.1022	0.8313	9.1728	2.0300	1325	**4825**
0.0625	0.9728	0.8144	4.9441	1.0081	4915	18675
0.03125	0.8980	0.8184	2.7087	0.2670	**18355**	71415

Stress and Error Results – Degrees of freedom of the models are added.



Graph of stress and error plot for the sample point.

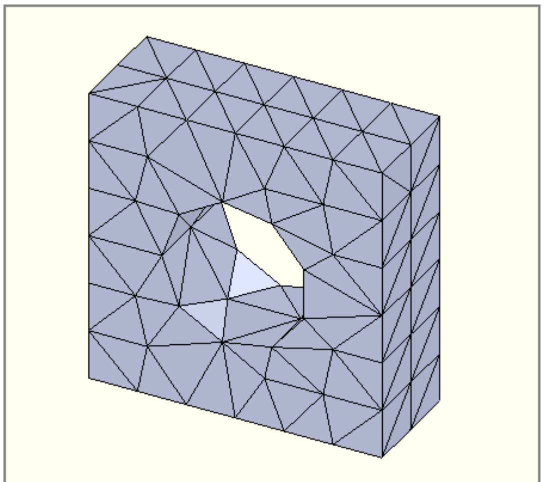
The 1st and 2nd order elements are both converging to the same stress value. The 2nd order models are getting to the end value much faster. The 2nd order result was obtained at 1/8" mesh size when the error was reported at 2%. The 1st order elements have not got there at 1/32" – and the reported error is above 2%. From the COSMOSWorks help files:

> It is highly recommended to use the **High** quality option for final results and for models with curved geometry. Draft quality meshing can be used for quick evaluation.

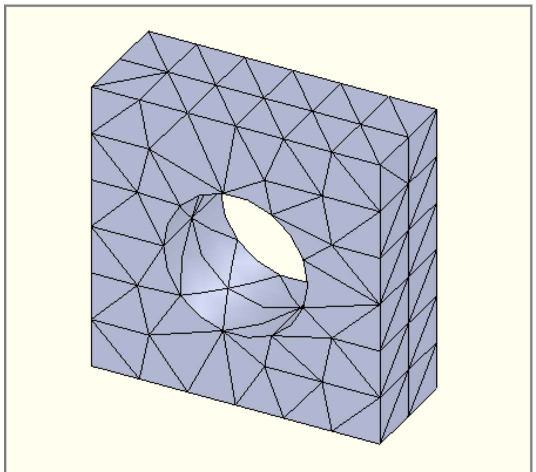
The degree of freedom of the model is related to the computer resources required to solve the problem. In this case, the 1st order model did not reach the result with a DOF of 18,000, but the 2nd order study got there by DOF = 4,800, a much better use of computer resources and users time.

Solid Models

The same mesh quality issues apply to 3D as to the previous 2D studies. Here is a part with a round hole. With a coarse mesh size, the 1st order model only slightly looks round. The second order results look much better.



1st order mesh on a block with a hole



Same mesh size – 2nd order elements

Why use 2nd Order Integration Elements?

1) ~~Because ABSA tells us to?~~
2) ~~Because that is the default Cosmos Designer Setting?~~
3) Because 2nd order elements do a better job of capturing the surface details.
4) Because 2nd order elements do a better job of calculating complex stresses.
5) Because 2nd order elements required fewer computer resources.