

All Models are Wrong



Structural analysis with Ansys Workbench

Third Edition

Wasseer Syed, P.E.

Copyright © 2025 Wasee Syed

All rights reserved. No portion of this book may be resold or reproduced in any form without permission from the author, except as permitted by U.S. copyright law.

For permissions contact: featips.help@gmail.com

*This work is dedicated to Ali ibn Abi Talib, the gate to the
city of knowledge*

Table of Contents

1. Why this book?	12
2. What is a mathematical model?	13
3. What are the factors that make every model unique?	15
3.1. Assumptions.....	15
3.2. Mathematical Equations	15
3.3. The Prediction	16
4. The most important ability to have as an engineering analyst.....	17
5. The framework for engineering analysis	17
6. Determine the objective(s) of the analysis	18
7. Determine the time allotted to achieve the objectives.....	20
8. Assess the capabilities and limitations of the analyst, the software, and the hardware	21
9. Determine the most efficient way to achieve the objective in the allotted time	22
10. Understand and balance the trade offs.....	22
10.1. Geometry Cleanup	22
10.2. Results accuracy vs Run time	23
10.3. Learning a new technique vs sticking to existing knowhow.....	23
11. Know and apply debugging techniques	24
12. Verify and present the results	25
13. Integrity, humility, and confidence.....	27
14. Types of Analyses.....	30
14.1. 2-Dimensional (2D) vs 3-Dimensional (3D) Analysis.....	30
14.2. Steady State vs Transient Analysis	30
14.3. Global, Local and Sub-Models	30
14.3.1. Global Model.....	30
14.3.2. Local Model	30
14.3.3. Submodel	31
15. Types of Structural Analyses.....	32
15.1. Linear vs Non-Linear Analysis.....	32
15.2. Static vs Dynamic Analysis	33
15.3. Linear Dynamic Analysis.....	34
15.3.1. Modal Analysis.....	34
15.3.2. Response Spectrum Analysis	34

15.3.3.	Harmonic Response Analysis	35
15.3.4.	Random Vibration Analysis	35
15.4.	Rigid vs Flexible Dynamic Analysis	35
16.	Finite Element Method (FEM).....	35
17.	Finite Element Mesh (FE Mesh)	36
17.1.	What is a Finite Element Mesh	36
17.2.	What is a mesh node?	36
17.3.	What is a degree of freedom?	37
17.4.	What is a shape function?	38
17.5.	What are mesh integration points	38
17.6.	Linear vs Quadratic elements.....	38
17.7.	Solid vs Shell vs Solid-Shell Elements	39
17.7.1.	Geometrical Characteristic	40
17.7.2.	Geometry Limitation.....	40
17.7.3.	Degrees of Freedom	40
17.7.4.	Geometry Preprocessing	40
17.7.5.	Meshing Freedom.....	40
17.7.6.	Meshing and Solution Time	41
17.7.7.	Through-Thickness Results	41
17.7.8.	Shear Locking.....	41
17.7.9.	Summary and Conclusion	42
17.8.	Element Types in Ansys	43
17.8.1.	Wedge15 and Pyramid13 Elements.....	46
17.9.	Conformal and Non-Conformal Meshes	48
17.9.1.	Introduction	48
17.9.2.	What is a Multi-Body Part?	48
17.9.3.	How do you create a Multi-Body part?.....	49
17.9.4.	The advantages of creating Multi-Body parts.....	49
17.9.5.	The disadvantages of creating Multi-Body parts	51
17.9.6.	Summary and Conclusion	52
17.10.	Ansys Mesh Methods Explained	52
17.10.1.	Introduction	52
17.10.2.	Mesh Categorizations	52
17.10.3.	Meshing Methods in Ansys.....	54

17.11.	Understanding Ansys Mesh Settings	58
17.11.1.	Understanding Sweep Mesh Settings	65
17.12.	How to verify mesh quality in Ansys Workbench?.....	70
17.12.1.	Mesh Metrics	70
17.12.2.	Summary.....	75
17.12.3.	Checking Mesh Metrics.....	76
17.13.	What is a mesh independence study?	77
17.14.	Is meshing an art or a science?	79
18.	Linear vs Non-Linear Analysis	79
18.1.1.	Introduction	79
18.1.2.	What is Non-Linearity?	80
18.1.3.	Linear vs Non-Linear Relationships.....	80
18.1.4.	Most relationships that exist in nature are non-linear	80
18.1.5.	Linear vs Non-Linear FEA	80
18.1.6.	Why would the stiffness of a structure not be constant?.....	81
19.	Ansys Geometry Stiffness Behavior Explained	83
20.	What is “time” in a Static Structural Analysis?.....	86
21.	How to decide when to run Non-Linear Analysis.....	87
22.	Material modeling and related topics	90
22.1.	Determining the yield strength of a ductile metal	90
22.2.	Rate-dependent vs Rate-Independent Plasticity.....	91
22.3.	Rate-Independent Plasticity	92
22.4.	Plasticity Models	92
22.4.1.	Isotropic vs Kinematic Hardening	93
22.4.2.	Bilinear vs Multilinear Hardening	95
22.4.3.	Non-Linear Hardening.....	95
22.4.4.	Which Plasticity Model to Pick?	95
22.5.	Failure Theories (Yield Criteria).....	96
22.5.1.	Maximum Shear Stress (Tresca) Criterion.....	96
22.5.2.	Maximum Distortion Energy (von-Mises) Criterion	97
22.5.3.	Comparing Tresca and von-Mises Yield Criteria	97
22.6.	Specifying a stress strain-curve in Ansys Mechanical.....	98
22.7.	Thermal de-rating of mechanical properties.....	98
22.8.	Elastic, Plastic and Total Strain.....	100

23. Failure Modes and Acceptance Criteria	101
23.1. What is a failure?	101
23.2. What is a failure mode?	101
23.3. What are some examples of failure modes?.....	101
23.4. What are acceptance criteria?	102
23.5. Stress Linearization	102
20.1. Plastic Collapse vs Limit load.....	105
24. Structural FEA theory – Step by Step Summary	109
25. Numerical Solvers and related topics	110
25.1. The general methods for solving engineering problems.....	110
25.1.1. Analytical Solution	110
25.1.2. Numerical Solution	110
25.1.3. Empirical Solution	110
25.2. Internal and External Forces	110
25.3. Force-Convergence Plots.....	112
25.3.1. Only relevant for a Non-Linear Analysis.....	112
25.3.2. A representation of The Newton-Raphson Method	113
22.3.1. Not a representation of the solver type.....	114
25.4. Direct vs Iterative Solvers.....	115
25.5. Load Controlled vs Displacement Controlled Models	116
25.5.1. What do we mean by load or displacement control?.....	116
25.5.2. Are they not interchangeable?	116
25.5.3. Which method is more suitable for structural analysis?.....	118
25.5.4. When to use displacement control for structural analysis?	118
25.5.5. A “hybrid” approach – best of both worlds	119
25.5.6. Conclusion	120
25.6. Weak Springs – Yes, or No?	120
25.6.1. When to turn on Weak Springs?.....	121
26. Contact Modeling in Ansys Workbench.....	121
26.1. Contact Settings	121
26.1.1. Scoping	122
26.1.2. Definition	123
26.1.3. Display	125
26.1.4. Advanced	126

26.1.5.	Geometric Modification.....	131
26.2.	Contact Formulations.....	131
26.2.1.	Penalty vs Lagrange Methods.....	131
26.2.2.	What is the Augmented Lagrange Method?.....	133
26.2.3.	Using the Stiffness and Tolerance Factors to aid in convergence	134
26.3.	MPC Contacts.....	134
26.3.1.	What are the advantages of MPC Contact?.....	134
26.3.2.	What are the Limitations of MPC Contact?	135
26.4.	Selecting the contact and target sides of a contact definition.....	136
26.5.	Contact Step Control – Activating and Deactivating Contacts	138
26.6.	Automatic Connections – Yes, or No?	140
26.6.1.	Consider the extensive set of options	140
26.6.2.	Go to contacts common to multiple bodies	141
26.6.3.	Create a Group	141
26.6.4.	Rename based on definition.....	142
26.6.5.	Hide all other bodies.....	142
26.6.6.	Use the Contact Tool	142
27.	Structural Dynamic Analysis.....	142
27.1.	Introduction	142
27.2.	Implicit vs Explicit Analysis.....	143
27.2.1.	Background on numerical analysis.....	143
27.2.2.	Implicit and Explicit Time Integration Techniques.....	145
27.3.	Transient Structural vs Rigid Dynamics Analysis in Ansys	147
27.4.	Modal Analysis	148
27.4.1.	What is Modal Analysis?.....	149
27.4.2.	Why do we perform Modal Analysis?.....	149
27.4.3.	What is natural frequency?	149
27.4.4.	What is damping?.....	150
27.4.5.	What are mode shapes?.....	151
27.4.6.	The governing equation of motion	151
27.4.7.	What is Simple Harmonic Motion?.....	152
27.4.8.	The Eigenvalue Problem	154
27.4.9.	How do we select the relevant natural frequencies?.....	155
27.4.10.	What is mode participation?	155

27.4.11.	Selecting the relevant modes?	156
27.4.12.	What about the displacements?.....	156
27.4.13.	Contacts in Modal Analysis.....	157
27.4.14.	Modal Analysis Summary.....	157
28.	Dynamic Amplification Factor.....	158
25.1.1.	What is a Dynamic Amplification Factor	158
25.1.2.	How is the Dynamic Amplification Factor Calculated?	158
25.1.3.	Dynamic Amplification Factor for a Pulse Load	160
29.	Tips, tricks, and efficient techniques	163
30.	General display and appearance.....	163
30.1.	Reset Layout.....	163
30.2.	Model lighting	164
30.3.	Results Legend transparency	164
30.4.	Rescale Annotations.....	165
30.5.	Working with labels (Results probes).....	166
30.6.	Simultaneous view ports.....	166
31.	Solving and Debugging.....	167
31.1.	Number of equilibrium iterations (NEQIT).....	167
31.2.	Stuck at “Preparing the mathematical model”	168
31.3.	Monitoring a run	170
31.4.	Dealing with convergence issues – “Element Distortion”	171
31.5.	Solver Pivot Warnings	174
31.5.1.	Introduction	174
31.5.2.	What is a Pivot?	174
31.5.3.	Examples of pivot related messages in Ansys.....	174
31.5.4.	What causes pivot related messages.....	175
31.5.5.	How to treat pivot warnings	176
31.6.	A guide to speeding up Ansys runs.....	177
32.	Post-Processing.....	179
32.1.	The solver files directory	179
32.2.	Extracting nodal information	179
32.3.	Freeing up disk space after a run	182
32.3.1.	What is the project scratch folder?.....	182
32.3.2.	How to locate the project scratch folder?	183

32.3.3.	Locating intermediate results files.....	184
33.	Miscellaneous	185
33.1.	Promote to Named Selection	185
33.2.	A guide to applying Bolt Pre-Tension	186
33.3.	Point Mass vs Remote Force	191
33.3.1.	What is a point mass?	191
33.3.2.	What is a remote fore?	192
33.3.3.	Which one to use for static structural analysis?	192
33.4.	Resetting App data to “fix” odd behavior	192
33.4.1.	What is Appdata?	193
33.4.2.	Resetting ANSYS Appdata	193

Part 1

Engineering Analysis

1. Why this book?

“Every vessel becomes narrow with what is placed in it, except for the vessel of knowledge, for it expands¹.” – Ali ibn Abi Talib

There is no shortage of high-quality texts on engineering analysis. One can acquire professionally written books on topics that cover the theoretical basis of numerical modeling techniques as well as the software packages available to perform such analyses. In addition to books, there is a variety of free (and paid) resource material available on the internet that will teach you the mathematics behind analysis and the practical steps needed to master software suites such as ANSYS and ABAQUS.

In my opinion, there are five aspects pertaining to engineering analysis:

1. Knowledge of the fundamental physics governing the problem at hand
2. Knowledge of the mathematical techniques required to model the physics
3. Knowledge of the software package in which to run the simulation
- 4. Insight that arises from a deep understanding of modeling and analysis**
- 5. Wisdom that is gained by understanding one’s abilities and limitations**

The resources that are widely available primarily (some exclusively) deal with the first three aspects. There is no doubt that these aspects are pre-requisites to succeeding as an engineering analyst. However, these three factors are **necessary**, but **not sufficient** conditions for excellence.

There are a number of “soft skills” that an analyst must acquire before they can do justice to the work that they are doing. By soft skills, we imply skills and abilities that may be difficult to recognize and quantify but play a significant role in the growth and development of an analyst.

These soft skills reflect one’s insight and wisdom as listed in points 4 and 5 above.

We must remember that wisdom is different from knowledge. A lot of knowledgeable people are unwise, and a lot of wise people have only basic knowledge.

In the first part of this book, we will make an effort to discuss the soft skills that can benefit an engineering analyst. We will look at the numerous ways in which these skills and abilities can enhance and enrich one’s experience and value as an analyst.

In Part 2 of this book, we will look at some of the technical details of structural analysis, numerical simulations and Ansys².

Finally, in part 3, we will touch upon some best practices for analysis in general and for Ansys Mechanical in particular.

¹ *Bihar-ul-Anwaar, Volume 1, Page 183*

² Ansys is one of the major companies specializing in building products geared towards scientific computation and engineering simulations. Ansys Mechanical is the structural analysis suite.