

Practical Aspects of Finite Element Simulation

A Study Guide

Table Of Contents

Table Of Contents	2
Foreword	8
About This Book	9
1. CAE-Driven Design Process	14
2. Analysis Types	16
3. Introduction To Finite Element Analysis	36
3.1 A Brief Summary	36
3.2 Opensource FEM Courses	57
4. What Is Needed To Run A Finite Element Analysis?	60
4.1 Basic Information Needed To Run A Finite Element Analysis	60
4.2 Modeling / Pre-Processing	60
4.3 Solution	63
4.4 Visualization / Post-Processing	63
5. Strategic Planning	65
5.1 Planning The Solution	65
5.2 Creating A Solution Checklist	72
5.3 Boundary Conditions And Load Cases	73
5.4 Linear Assumption	77
6. Common Mistakes And Errors	79
6.1 Modeling And Visualization	79
6.2 Errors Within Organizations	82
7. Consistent Units	85
7.1 Equations Used To Help Determine Consistent Units	85

Table Of Contents

8. Geometry In HyperMesh	87
8.1 HyperMesh Geometry Terminology.....	87
8.2 Geometry Cleanup	90
8.3 Geometry Creation And Editing.....	96
8.4 Importing CAD Geometry.....	98
8.5 Geometry FAQ's.....	99
8.6 Recommended Tutorials And Videos.....	102
8.7 Student Racing Car Project - Introduction And CAD Related Aspects	106
9. Introduction To Meshing.....	109
9.1 Why Do We Carry Out Meshing?	109
9.2 Types Of Elements	110
9.3 How To Decide The Element Type.....	110
9.4 Can We Solve The Same Problem Using 1D, 2D And 3D Elements	113
9.5 How To Decide Element Length.....	114
9.6 How To Start Meshing	114
9.7 Meshing Techniques.....	116
9.8 Meshing In Critical Areas.....	117
9.9 Mesh Display Options.....	118
9.10 Understanding Element Behavior.....	120
9.11 Element Selection.....	124
9.12 Mesh Density And Solution Convergence	125
10. 1D Meshing With HyperMesh	131
10.1 When To Use 1D Elements	131
10.2 Special Features Of Beam/Bar Elements.....	132
10.3 Rigid Elements	134
10.4 Fasteners	136
10.5 1D Element Creation	138
10.6 Connectors In HyperMesh.....	157
10.7 Learn More About Connectors.....	159
10.8 1D Meshing Tutorials And Videos.....	175
10.9 Student Racing Car Project - 1D Meshing	178

Table Of Contents

11. 2D Meshing With HyperMesh	180
11.1 When To Use 2D Elements.....	180
11.2 Family Of 2D Elements.....	184
11.3 Thin Shell Elements.....	187
11.4 Effect Of Biasing In The Critical Region.....	192
11.5 Symmetric Boundary Conditions	193
11.6 Geometry Associative Mesh	198
11.7 How Not To Mesh.....	199
11.8 Creating 2D Elements in HyperMesh.....	203
11.9 Automeshing.....	204
11.10 Shrink Wrap Meshing.....	206
11.11 Meshing FAQ's	207
11.12 2D Meshing Tutorials And Videos.....	212
12. 3D Meshing with HyperMesh	215
12.1 When To Use 3D Elements	215
12.2 3D Element Types.....	216
12.3 Dofs For Solid Elements.....	216
12.4 Tetra Meshing Techniques	216
12.5 Brick Meshing.....	219
12.6 Tips For Brick Meshing.....	221
12.7 How Not To Mesh.....	221
12.8 Creating 3D Elements Using HyperMesh.....	224
12.9 Tutorials And Videos.....	228
13. Element Quality And Checks.....	234
13.1 Compatibility And Mechanisms	234
13.2 General Element Quality Checks.....	238
13.3 2D Quality Checks	240
13.4 Other Checks For 2D Meshing.....	242
13.5 Quality Checks For Tetra Meshes	245
13.6 Other Checks For Tetra Meshes	246
13.7 Brick Mesh Quality Checks	248

Table Of Contents

13.8 Other Checks For Brick Meshes	249
13.9 Mesh Check Tools In HyperMesh	250
13.10 Mesh Check Tools Tutorials And Videos.....	254
13.11 Student Racing Car Project - Mesh Quality.....	257
14. Linear Elastic Material Information.....	260
14.1 Hooke's Law And Two Constants	260
14.2 Generalized Hooke's Law And 36 Total Constants In The Equation	261
14.3 Material Classification.....	261
14.4 Material Properties.....	262
14.5 Linear Elastic Material And Property Tutorials And Videos.....	263
14.6 Student Racing Car Project - Material Definition.....	267
15. Boundary Conditions And Loads.....	274
15.1 Boundary Conditions	274
15.2 How To Apply Constraints.....	286
15.3 Symmetry	295
15.4 Creating Loadsteps In HyperMesh	299
15.5 Discussion On AUTOSPC In OptiStruct	299
15.6 Recommended Tutorials And Videos	305
15.7 Student Racing Car Project - Boundary Conditions	308
16. Linear Static Analysis	311
16.1 Linear Static Analysis	311
16.2 Linear Static Analysis Example Using HyperMesh	312
16.3 Linear Static Analysis Setup Using HyperMesh.....	314
16.4 Linear Static Analysis Tutorials And Videos	322
16.5 Example: Analysis Of A Rotating Disc.....	325
16.6 Student Racing Car Project: Linear Static Analysis	332
17. Modal Analysis.....	333
17.1 Introduction	333
17.2 Example: Modal Analysis	336

Table Of Contents

17.3 Example: Modal Analysis With Constraint Model.....	340
17.4 Tips & Tricks Regarding Post-Processing.....	341
18. Linear Buckling Analysis	345
18.1 Introduction.....	345
18.2 Elastic Buckling	346
18.3 Linear Buckling Analysis With OptiStruct.....	347
18.4 Example: Linear Buckling Analysis Of A Beam Structure.....	349
18.5 Example: Wing Linear Buckling Analysis.....	354
18.6 Example: Buckling With Gravitational Load	356
18.7 Linear Buckling Analysis Tutorials And Videos	360
19. Nonlinear Analysis.....	361
19.1 Introduction.....	361
19.2 Comparison Of Linear And Nonlinear FEA	361
19.3 Types Of Nonlinearity	362
19.4 Stress-Strain Measures For Nonlinear Analysis	366
19.5 Essential Steps To Start With Nonlinear FEA.....	367
19.6 General Procedure For Nonlinear Static Analysis Project	368
19.7 Nonlinear Analysis With OptiStruct & RADIOSS - Overview	369
19.8 Nonlinear Solution Sequences in OptiStruct	371
19.9 Types Of Nonlinearity.....	372
19.10 Modeling Elastic-Plastic Material In OptiStruct 13.0	375
19.11 EXAMPLE: TENSILE TEST (elastic-plastic).....	381
19.12 Contact Analysis in OptiStruct 13.0	400
19.13 Auto-Contact Manager	425
19.14 Contact Modeling in Structural Simulation – Approaches, Problems and Chances.....	431
19.15 Nonlinear Analysis Tutorials And Videos	442
20. Post-Processing.....	443
20.1 How To Validate And Check Accuracy Of The Result	443
20.2 How To View And Interpret Results.....	443
20.3 Post-Processing In HyperView.....	448
20.4 Stress Calculation And Output In Optistruct.....	455

Table Of Contents

20.5 Special Tricks For Post-Processing.....	468
20.6 Interpretation Of Results And Design Modifications	476
20.7 CAE Reports	481
20.8 Post-Processing Tutorials and Videos	483
20.9 Student Racing Car Project: Post-Processing.....	485
21. Getting Started With HyperWorks	489
21.1 The HyperWorks Desktop Graphical User Interface	489
21.2 Tutorials And Videos	497
22. Preparing For An Interview	498

Foreword

Simulation has come a long way since it was generally adopted in Aerospace and Automotive industries in the 1970's. On the other hand, we still see archaic terminology such as "card image" when today's users may have never seen or used a "card" punch.

Engineers are normally conservative and somewhat slow to change which is a good thing for the safety aspects of the products they design. But we live in an ever changing and accelerating technological era. Today, simulation is used in every industry that designs and manufactures products from toothpaste tubes to rocket engines. The demand for engineers in the simulation field has never been higher. The advance of computing is relentless. Today's mobile devices are more powerful and can store more information than the supercomputers of the early days of simulation. The pace of computing continues to grow at an exponential rate even though a receding growth rate has been predicted many times. The only thing that is not changing is the basic physics behind the simulation computations. So where are we going with all of this advance in capability and demand?

The level of detail in the simulation models continues to increase with fewer and fewer assumptions required to get answers in a reasonable time. As the models become more complex the requirements for assembling models, analysis of the results, and storing the volumes of input and output are increasing. Altair is working to bring the future of simulation to engineers in both an evolutionary and revolutionary way. We continue to modernize our tradition tools for ease of use, large model handling, and efficiency and at the same invent new ways of working for the future generations of engineers.

The challenge for the future is to make the complex simple by reducing the requirements put on the user to get into the details unless necessary or desired. The simulation process can be more systematic and guided versus more of an art as it is sometimes today. Two engineers solving the same problem should be able to get similar results by establishing processes that are repeatable. More realistic representation of reality can be achieved by stochastic analysis where the variation of real products is a standard part of the simulation process. Data and processes can be shared within teams and across teams. Finally with advance in technology we will see a return to remote computation or as it is called today cloud computing which will include high speed 3D graphics. Altair is working in all of the directions described above to bring the best value and experience to the engineering community worldwide.

In the document that follows, you will see elements of where we are going. But the reality is there is a lot to learn and the knowledge you need to successfully apply simulation is quite vast. Good luck in your endeavors and we hope we can be of value to you now and throughout your career.

James E. Brancheau

CTO, Altair Engineering

About This Book

Over the years Altair has matured into a solver company with OptiStruct (linear and non linear implicit, structural optimization), RADIOSS (non-linear explicit Finite Element Analysis), solidThinking Inspire (structural optimization), MotionSolve (multibody simulation), AcuSolve (Computational Fluid Dynamics or CFD), and FEKO (electro-magnetic field simulation).

Altair has always extended its hands to academia – in the past mainly with regards to modeling (pre-processing) and visualization (post-processing). Today, Altair's CAE offering is more complete and suitable to universities. Thus, we are very grateful that many universities adopted HyperWorks in their engineering programs in order to train students in the Simulation-Driven Design and Optimization Process.

The **HyperWorks Student Edition 13.0** in combination with our learning and teaching program target the needs of teachers and students.

"It is exciting news that the student version of HyperWorks is now available," said Dr. Wei Chen, Wilson-Cook Professor in Engineering Design at Northwestern University. "This program truly demonstrates Altair's commitment to enhancing students' learning experience outside the classroom so that they can reinforce their engineering knowledge and classroom instruction."

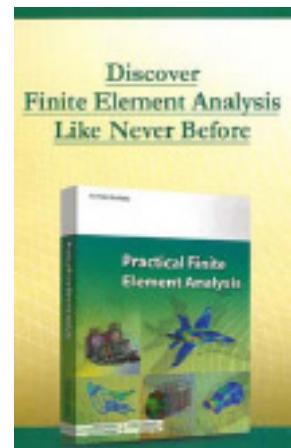
Still, we realized that a dedicated CAE book including some best practice tips were missing. This was the moment when we decided to publish our CAE expertise in a book addressing questions typical raised by students.

At that time it so happened that we learned about the book by Nitin S. Gokhale and his team with the title

Practical Finite Element Analysis (from Finite To Infinite)

An amazing book with the focus on what we were targeting at. Nitin S. Gokhale and his team immediately agreed to merge their CAE book and Altair's simulation training material into this Study Guide.

Even though it is the 3rd Edition of this collaborative work, it is by far not complete. The book still is "Work in Progress". More topics such as Crash Simulation, Thermal Analysis, NVH, CFD, and Electro-Magnetic Field Simulation will be added step by step.



This book intends to provide basic information to help the beginner Finite Element Analysis (FEA) modeler as well as the novice HyperWorks user. to get started.

And we know that becoming familiar with a (new) CAE system such as HyperWorks is somewhat comparable to learning, for instance, a new language. At the same time, we all face a major challenge: not having enough time. This challenge is universal. This is because we are educated to constantly increase the pace of our lives and work.

Hence, learning new technologies must be as easy (and also as comfortable) as possible. Otherwise, as practice shows, making the first step is very difficult.

At the beginning there are many questions, such as:

- Where to start?
- Which "rules" are important?
- Where to learn?
- When to learn?
- etc.

So how does one become familiar with FEA and also become a proficient HyperWorks user?

In order to support your endeavors in learning more about FEA and HyperWorks, we offer (in addition to this book):

- An extended set of E-Learning material such as (free) webinars and videos
- A comprehensive set of tutorials and manuals (e.g. HyperMesh Basic, OptiStruct Basic, free Study Guides, etc.),
- Highly discounted seminars at colleges or at Altair facilities,
- Best practice Tips and Tricks,
- Academic User Meetings,
- Academic Blog (our HyperWorks knowledge database which focuses on and addresses the needs of students, teachers and researchers),
- Moderated Support Forum,
- Academic Newsletters,
- and more

We are confident that this “infrastructure” will help you get up to speed quickly.

But nevertheless, we would be more than pleased to incorporate your own summary (article or chapter) in this book. No question, your contribution will be acknowledged accordingly – just as we do with all the contributions we have received so far.

So please stay tuned, enjoy the latest edition of this book and let us know whether it helped you getting started with CAE.

Best regards

Dr. Matthias Goelke

On behalf of “The HyperWorks University Team” in cooperation with Finite To Infinite

Some Remarks About Altair's Academic Program

An ideal supplement to this book is the **HyperWorks Student Edition** which allows you to study and practice the various topics addressed in this book.

More information about the HyperWorks Student Edition is available on the

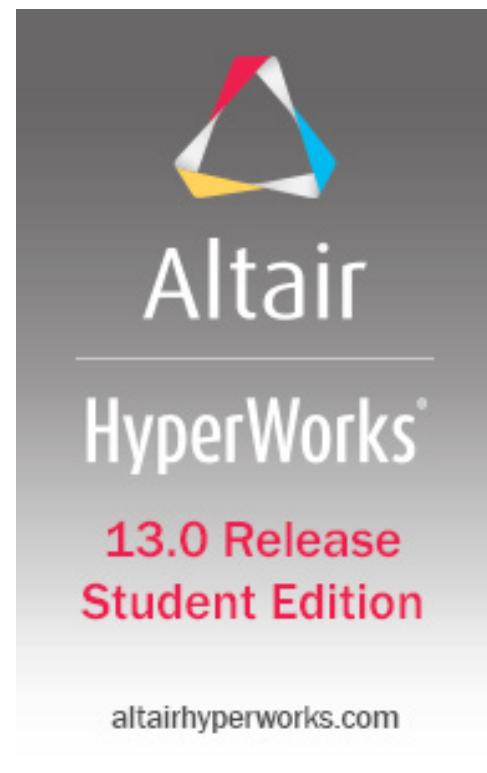
Altair Academic Blog (www.altairuniversity.com).

On the **Academic Blog** and the

Academic Training Center (www.training.altairuniversity.com)

you will also find more material about:

- how to get started
- tutorials
- project ideas
- software news
- tips & tricks
- e-learning material
- academic events
- student challenges and sponsoring
- career opportunities
- and much more ...



If you need help cracking any technical queries pertaining to Altair's software suite HyperWorks, please feel free to post your questions on the moderated **Academic Support Forum** (www.altairuniversity.com/support-forum/). The image of the Support Forum shown below is a “cutout” of the many different categories hosted there.

Solver			
	RADIOSS Moderators: Matthias Goelke, Chayan Basak, Manjunath K, Ashraf Hussain, Rahul Ponginan, PrakashPagedala	Topics: 58 Posts: 408	Last post by PrakashPagedala in Re: Message ID 205 and 207 on October 1, 2013, 03:42
	OptiStruct Moderators: Matthias Goelke, Chayan Basak, Manjunath K, Ashraf Hussain, Rahul Ponginan, PrakashPagedala	Topics: 95 Posts: 491	Last post by Rahul Ponginan in Re: Error # 1310 on September 23, 2013, 00:15
	AcuSolve Moderators: Matthias Goelke, Chayan Basak, Manjunath K, Ashraf Hussain, Rahul Ponginan, PrakashPagedala	Topics: 13 Posts: 53	Last post by Rahul Ponginan in Re: How does AcuSolve handle wall roughness? on September 30, 2013, 08:21
	MotionSolve Moderators: Matthias Goelke, Chayan Basak, Manjunath K, Ashraf Hussain, Rahul Ponginan, PrakashPagedala	Topics: 4 Posts: 25	Last post by sankar_venkatesh in Re: Eigen frequency output as expression on July 4, 2013, 11:22

Hyperworks For Teaching

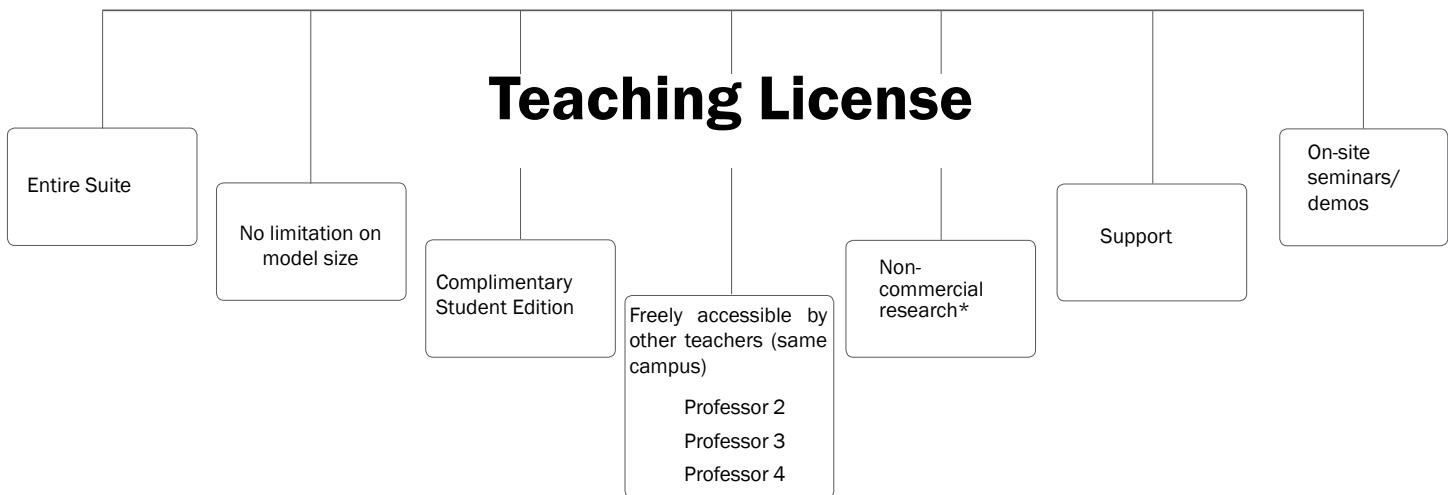
Leading universities across the globe are using HyperWorks computer aided engineering (CAE) simulation software for teaching and research in the fields of:

- Structural analysis
- Computational fluid dynamics (CFD)
- Optimization
- Multi-Body Dynamics (MBD)
- Electro-Magnetic Field simulation (EM)
- Numerical methods & programming

and much more!

Altair has commercial expertise to share with the academic community. By including real life scenarios in your teaching material, Altair can help you add value to your engineering design courses.

Our unique licensing system allows universities to use the entire HyperWorks suite in a very flexible and cost efficient way.



*Results may be used for marketing, training and demo purposes

Please let us know your requirements by sending an e-mail notification to

altairuniversity@altair.com

We are more than happy helping and assisting you with your teaching activities.

Acknowledgement

A very special **THANK YOU** goes to:

Rahul Ponginan (Altair India) who thoroughly reviewed this Study Guide. Rahul's feedback significantly improved its quality.

- Rajneesh Shinde, Nelson Dias, Srirangam R. Srirangarajan, Prakash Pagadala (Altair India)
- Elizabeth White, Sean Putman, David Schmueser, John Brink, James Brancheau, Lena Hanna, Ralph Krawczyk, Chad Zamler, Jeff Brennan, and the entire HyperWorks Documentation Team (Altair USA)
- Tony Gray (Altair Australia)
- Hossein Shakourzadeh (Altair France)
- Baljesh Mehmi, Gareth Lee, Nicola Turner (Altair UK)
- Jan Grasmannsdorf, Thomas Lehmann, Sascha Beuermann, Debdatta Sen, Kristian Holm, Christian Steenbook, Patrick Zerbe, Marian Bulla, Juergen Kranzeder, Bernhard Wiedemann, Carolina Penteado, Jacob Tremmel and Moritz Günther (Altair Germany)
- Markus Kriesch and Andre Wehr (Universität der Bundeswehr München / Germany)
- Professor Rolf Steinbuch (Reutlingen University; Germany) for the chapter "Contact Modelling in Structural Simulation – Approaches, Problems and Chances"

Disclaimer

Every effort has been made to keep the book free from technical as well as other mistakes. However, publishers and authors will not be responsible for loss, damage in any form and consequences arising directly or indirectly from the use of this book.

© 2015 Altair Engineering, Inc. All rights reserved. No part of this publication may be reproduced, transmitted, transcribed, or translated to another language without the written permission of Altair

Engineering, Inc. To obtain this permission, write to the attention Altair Engineering legal department at:
1820 E. Big Beaver, Troy, Michigan, USA, or call +1-248-614-2400.

Trademark and Registered Trademark Acknowledgments

Listed below are Altair® HyperWorks® applications. Copyright© Altair Engineering Inc., All Rights Reserved for:

HyperMesh® 1990-2015; HyperCrash™ 2001-2015; OptiStruct® 1996-2015; RADIOSS® 1986-2015; HyperView® 1999-2015; HyperView Player® 2001-2015; HyperStudy® 1999-2015; HyperGraph® 1995-2015; MotionView® 1993-2015; MotionSolve® 2002-2015; HyperForm® 1998-2015; HyperXtrude® 1999-2015; Process Manager™ 2003-2015;

Templex™ 1990-2015; Data Manager™ 2005-2015; MediaView™ 1999-2015; BatchMesher™ 2003-2015;
TextView™ 1996-2015; HyperMath™ 2007-2015; ScriptView™ 2007-2015; Manufacturing Solutions™ 2005-2015;

HyperWeld™ 2009-2015; HyperMold™ 2009-2015; solidThinking™ 1993-2015; solidThinking Inspire™ 2009-2015;
Durability Director™ 2009-2015; Suspension Director™ 2009-2015; AcuSolve™ 1997-2015; and AcuConsole™ 2006-2015

1 CAE-Driven Design Process

This chapter includes material from the book “**Practical Finite Element Analysis**” as well as information from the HyperWorks Help Documentation. Additional material was added by Matthias Goelke and Jan Grasmannsdorf.

The “CAE-Driven Design Process” has gained significant attraction in most industries such as aerospace, automotive, biomedical, consumer goods, defense, energy, electronics, heavy industry, and marine throughout the last years. There are many reasons for the overall acceptance of CAE as simulation has proven to help with:

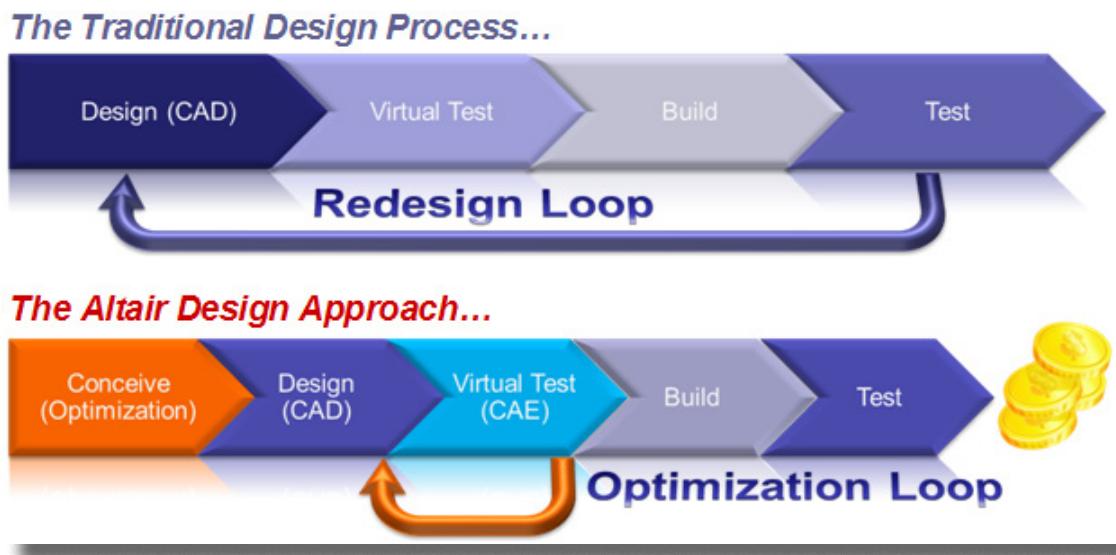
- New and inspiring designs
- Products with better quality (e.g. increased material efficiency where less material = lighter designs)
- Designing faster (i.e. due to shortened development cycles and a reduction in the number of prototypes by minimizing “Trial and Error” attempts)

In other words, simulation saves time, reduces costs, and essentially strengthens the competitiveness of companies, thus strengthening their market position.

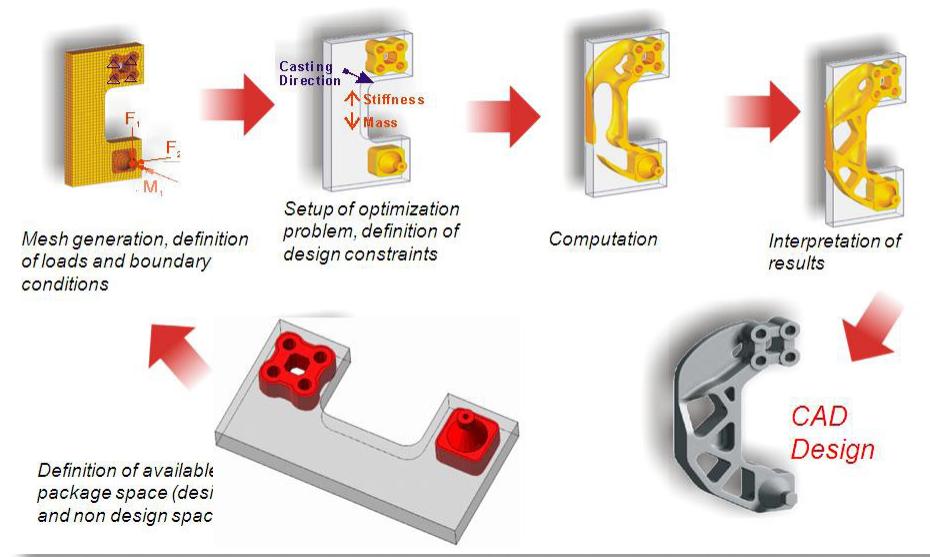
When doing an analysis, we always target optimum designs, but the methods and tools we use in achieving the optimum design makes a difference. Still, in many “places” the design process is a trial-and-error process which depends on the selection of the initial design. This is not a well-established process in terms of how the design evolves and depends on the engineer’s prior experience. Due to these challenges reaching the best design is not guaranteed.

To overcome these hurdles, numerical optimization is used to search for and determine the optimum design.

With the Altair Design Approach, a “concept” optimization step is employed early in the design process which delivers a conceived design proposal.



Starting the CAE process with an optimized concept design sounds contradictory. How does one start with something being optimized while not really knowing how to start?



Without getting lost in details – the principal working scheme (depicted in the figure above) is as follows:

1. Define the maximum package/design space. If needed exclude areas/regions (i.e. non-design space)
2. Finite element mesh the structure (design and non-design area)
3. Assign material properties
4. Apply loads and constraints
5. Specify the objective of the optimization problem (e.g. use minimum weight/volume only but make sure that certain responses such as displacements, do not exceed specified threshold values)

This optimization delivers a design concept which answers questions such as where to remove material, where to place (stiffening) ribs, etc. After the optimization results have been interpreted and smoothed (in the CAD system) an analysis is carried out to verify the optimized components performance. Based on these results , another optimization step may follow, addressing questions such as how to change local geometry in order to reduce stress peaks or how thick do the stiffening ribs really need to be.

By following this design process, unnecessary redesign loops are eliminated leading not only to shortened design cycles but also to more competitive products.



Racing Car of the Monash Motorsport team

“Optimization of a structural part is all about efficient material use. Using OptiStruct takes the question of material distribution in a design out of the equation and tells you exactly what you need to do and where the material needs to be. If you use OptiStruct, you can skip all that messing around with different options, because it gives you the best options straight away.”

(Mark Stroud, Monash Motorsport Team)

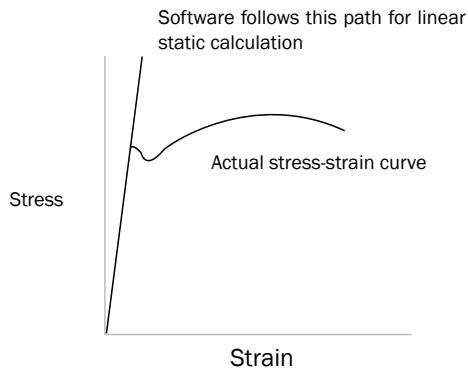
2 Analysis Types

This chapter includes material from the book “**Practical Finite Element Analysis**” as well as information from the HyperWorks Help Documentation. Additional material was added by Matthias Goelke and Jan Grasmannsdorf.

The term CAE (Computer Aided Engineering) includes the following types of analyses:

- | | |
|---------------------------|---------------------|
| 1) Linear static analysis | 6) Fatigue analysis |
| 2) Nonlinear analysis | 7) Optimization |
| 3) Dynamic analysis | 8) CFD analysis |
| 4) Buckling analysis | 9) Crash analysis |
| 5) Thermal analysis | 10) NVH analysis |

1) Linear Static Analysis



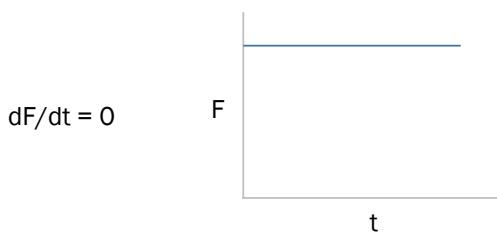
Linear

Linear means straight line. In linear analysis, the FE solver will therefore always follow a straight line from base to deformed state. As an example in terms of linear material behaviour, $\sigma = E \epsilon$ is the equation of a straight line ($y = m x$) passing through the origin. “E”, the Elastic Modulus, is the slope of the line and is a constant. In real life after crossing the yield point, the material follows a nonlinear curve but solvers follow the same straight line. Components are broken into two separate pieces after crossing the ultimate stress point, but software based linear analysis never shows failure in this fashion. It shows a single unbroken part with a red color zone at the location of the failure. An analyst has to conclude whether the component is safe or has failed by comparing the maximum stress value with yield or ultimate stress.

Static

There are two conditions for static analysis:

- 1) The force is static i.e. there is no variation with respect to time (dead weight)



2) Equilibrium condition \sum forces (F_x, F_y, F_z) and \sum Moments (M_x, M_y, M_z) = 0.

The FE model must fulfil this condition at each and every node. The complete model summation of the external forces and moments is equal to the reaction forces and moments.

The basic finite element equation to be solved for structures experiencing static loads can be expressed as:

$$\mathbf{K} \mathbf{u} = \mathbf{P}$$

where K is the stiffness matrix of the structure (an assemblage of individual element stiffness matrices). The vector u is the displacement vector, and P is the vector of loads applied to the structure. The above equation is the equilibrium of external and internal forces.

The stiffness matrix is singular, unless displacement boundary conditions are applied to fix the rigid body degrees of freedom of the model.

The equilibrium equation is solved either by a direct or an iterative solver. By default, the direct solver is invoked, whereby the unknown displacements are simultaneously solved using a Gauss elimination method that exploits the sparseness and symmetry of the stiffness matrix, K, for computational efficiency.

Once the unknown displacements at the nodal points of the elements are calculated, the stresses can be calculated by using the constitutive relations for the material. For linear static analysis where the deformations are in the elastic range, i.e.: the stresses, σ , are assumed to be linear functions of the strains, ϵ , Hooke's law can be used to calculate the stresses. Hooke's law can be stated as:

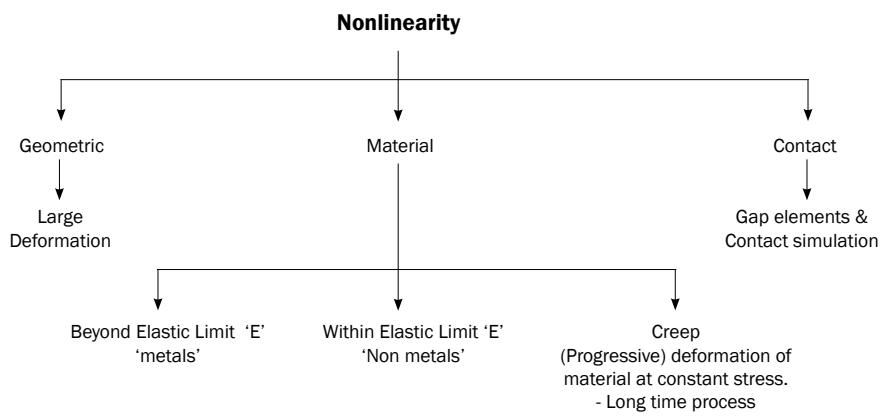
$$\sigma = C \epsilon$$

with the elasticity matrix C of the material. The strains are a function of the displacements.

Please note that none of the terms in the equation is dependent on time or displacement. This is why it is called, as we have learned before, a linear static analysis.

Practical Applications: A linear static analysis is the most commonly used analysis. All aerospace, automobile, offshore and civil engineering industries perform linear static analyses.

2) Nonlinear Analysis



Nonlinear Analysis Can Mean:

A nonlinear analysis is performed when we need to consider

A. Material based non linearity: this is related to for instance, nonlinear elastic, elastoplastic, viscoelastic, and/or viscoplastic behavior. The Force (stress) vs. displacement (strain) curve is nonlinear (polynomial).

B. Geometric nonlinearity: In real life, the stiffness [K] is a function of displacement [d] (remember: for linear analysis [K] is constant, independent of [d]). This means in a geometric nonlinear analysis, the stiffness K is re-calculated after a certain predefined displacement. As an example, think of a buckling case. The stiffness of a geometry changes dramatically when or after buckling has occurred. The new “deformed” geometry must now be taken into account, in order to get reasonable (realistic) results.

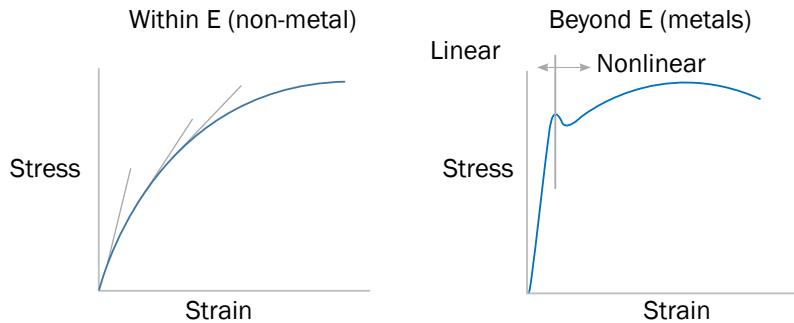
C. Contact nonlinearity / boundary nonlinearity: In contact analysis, the stiffness K also changes as a function of displacement (when parts get into contact or separate)

Note: Nonlinear analysis deals with true stress and strain (unlike engineering stress and strain in linear static analysis)

In his script about NONLINEAR FINITE ELEMENT ANALYSIS, Doc. Ing. Vladimír Ivančo, provided a comprehensive summary on the consequences of nonlinear structural behavior:

- a) The principle of superposition cannot be applied. Thus, for example, the results of several load cases cannot be combined. Results of the nonlinear analysis cannot be scaled.
- b) Only one load case can be handled at a time.
- c) The sequence of application of loads (loading history) may be important. Especially, plastic deformations depend on the manner of loading. This is a reason for dividing loads into small increments in nonlinear FE analysis.
- d) The structural behavior can be markedly non-proportional to the applied load.
- e) The initial state of stress (e.g. residual stresses from heat treatment, welding, cold forming etc.) may be important

A. Material Nonlinearity



The stress-strain diagram, along with the hardening* rule for the material, is required as input data.

*As a side note –what does strain hardening mean? When loading is carried beyond the yield point, the load needs to increase for additional strain to occur. This effect is called strain hardening, and it is associated with an increased resistance to slip deformation at the microscale (for polycrystalline materials). Eventually, the stress-strain curve reaches a maximum at the point of ultimate stress. For many materials, the decrease in the cross-sectional area of the specimen is not readily visible to the naked eye until this limit point is passed (from John Dolbow; <http://people.duke.edu/~jdolbow/>)

Metallic nonlinearity applications: Automobile, aerospace, ship industries. An analysis is done to know the exact value of stress or strain when it crosses the yield point. For low cycle fatigue analysis, this data is considered as input for the strain life approach.

Non-metallic nonlinearity applications: Automobile, aerospace industry, analysis of rubber, plastic, asbestos, fiber components.

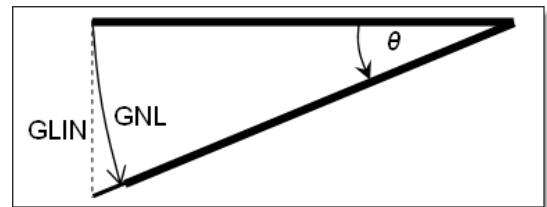
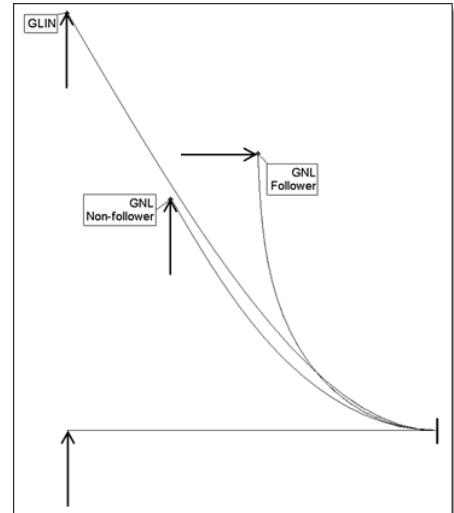
Creep - At elevated temperatures, even a small magnitude force, if kept applied over a long time period (for months and years), would cause failure. Applications – nuclear / thermal power plants, civil engineering etc.

B. Geometric Nonlinearity

In geometric linear analysis all deformations and rotations are small (infinitesimal). As a general rule, displacements of say 5% of the model dimension and rotations up to 5 degrees can be treated as small. Rotations are trickier. A rotating body seems to get bigger linearly under deformation even if defined as rigid. Loads stay in the undeformed coordinates and simply move along the axis they are defined in.

In geometric nonlinear analysis, displacements and rotations are large (finite). The magnitude of a force actually matters. Changes in magnitude of a load can change convergence behavior considerably. Also the direction of a force needs to be controlled. Forces may follow the deformation or keep their direction. This can be controlled thru the choice of coordinate systems (in geometric nonlinear analysis there are moving and fixed coordinate systems).

The images depict two examples of these differences: A cantilever beam solved with small displacements, large displacements with a follower force, and large displacements without a follower force. The image below depicts a simple rigid rotated by an angle solved with small and finite rotations.

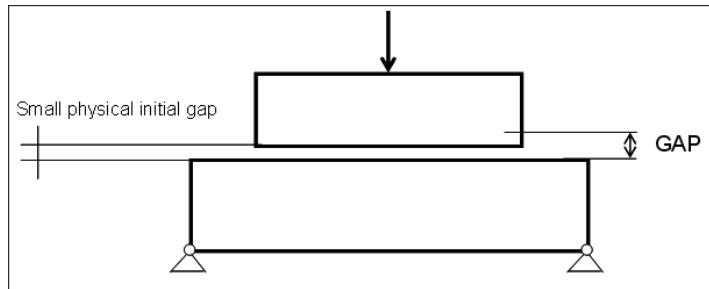


The question on “*how to decide on geometric nonlinearity*” was also discussed on imechanica.org. For your convenience, we included an interesting comment / statement by Peyman Khosravi (link last viewed on May 28, 2015)

“I worked on Geometrically Nonlinear (GN) structures for a couple of years and I had the same question in my mind. At the beginning I would think it is only the amount of displacement/rotation which defines whether it is a GN case i.e. if it is small it is a linear case and if it is not, it is a GN case. However, GN cases are not limited to large displacement/rotations. A cantilever beam with an end force can be assumed liner, if for example the end displacement is up to 1/10 of its length. I mean you can get a good approximation by linear analysis up to that range of displacement. But a shallow simply-supported arch with the same length under vertical downward load at its top point is highly nonlinear even when the displacements are still small, before going to snap-through and change of its shape. Another example is buckling of shells which happen with small initial displacements but under a very nonlinear behavior. Long story short, I believe it needs a lot of experiment to be able to judge the case by just looking at it. However if the case is somehow related to buckling and bifurcation analysis, it is most probably a GN case because bucklings (and snap-throughs) happen in small displacements and deal with membrane or axial loads. You can look at the structure and guess if the deformation is mostly driven by bending or an axial loads. If it is by axial loads it is mostly a GN case and if it is bending, it is GN when deformations are large. Of course the best way is to do GN analysis and compare with linear analysis and if there is a big difference it is GN”

C. Contact

This type of nonlinear analysis is used to simulate the physical gap between two parts (bearing and shaft, press fit between two cylinders, etc.), parts placed on top of each other, parts that may collide i.e. come in contact with each other etc



Nonlinear Quasi Static (small deformation)

This solution sequence uses small deformation theory, similar to the way it is used with Linear Static Analysis. Small deformation theory means that strains should be within linear elasticity range (some 5 percent strain), and rotations within small rotation range (some 5 degrees rotation). This also means that there is no update of gap/contact element locations or orientation due to the deformations – they remain the same throughout the nonlinear computations. The orientation may change, however, due to geometry changes in optimization runs. Inertia relief is also possible. Presently, the sources of nonlinearity include CONTACT interfaces, GAP elements, and MATS1 elastic-plastic material.

Nonlinear Quasi Static (large deformation)

Large displacement nonlinear static analysis is used for the solution of problems wherein the load-response relationship is nonlinear and structural large displacements are involved. The source of this nonlinearity can be attributed to multiple system properties, for example, materials, geometry, nonlinear loading and constraint. Currently, in OptiStruct the following large displacement nonlinear capabilities are available, including large strain elasto-plasticity, hyperelasticity of polynomial form, contact with small tangential motion, and rigid body constraints.

3) Dynamic Analysis

a) Linear Dynamics

Describes those systems in which forces increase linearly with parameters such as position and velocity. Perhaps the best known linear system is a mass oscillating on a spring, or the “simple harmonic oscillator.” In this situation, force on the mass increases linearly with displacement by a factor of “k,” the spring constant. A graph of the potential energy of this system is parabolic, since $F(x) = -dU/dx$.

A particle of mass m oscillating in the potential well of this system will have an angular frequency of $\omega = \sqrt{\frac{k}{m}}$. The system becomes more complex when damping or a driving force is added. Damping alone will cause the particle to sit down in the potential well or “attractor.” When the system is forced, the system will oscillate at one frequency determined by the relative strengths of the forcing and damping. If the forcing is too weak or too strong, the system may oscillate at the forcing frequency. In this case, the free movement of the system is essentially drowned by the forcing and damping and the amplitude of oscillation is weak. At some frequency, however, the system will resonate where maximum oscillating amplitude occurs. Again, in the linear system, this resonant frequency is unique and is determined by the forcing, damping, and natural frequencies of the system. In this driven and damped linear system, periodic inputs result in periodic outputs. If there is error in a measurement, this error will increase linearly as the system progresses.

b) Nonlinear Dynamics

Describes those nonlinear systems where one or more “forcing elements” does not vary linearly with space parameters. For example, if the spring coefficient in the spring system described before varied with displacement, then the spring force would vary with the square of displacement.

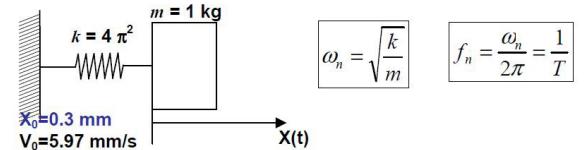
Although linear systems make for pretty equations and an efficient summary of behavior, nonlinear systems seem to pervade real natural systems. Friction forces, damping elements, resistive elements in circuits – these and many other factors often vary

in a nonlinear fashion. As a result, the differential equations describing these systems involve very messy solutions that can only be solved numerically. Even if there were analytic solutions to these systems the behavior in some cases would be difficult or impossible to predict due to the exponential increase in error

Static Versus Dynamic System

To help us understand what a dynamic system is it is interesting to compare it with a static system. There are two basic aspects that make the dynamic systems differ from static systems:

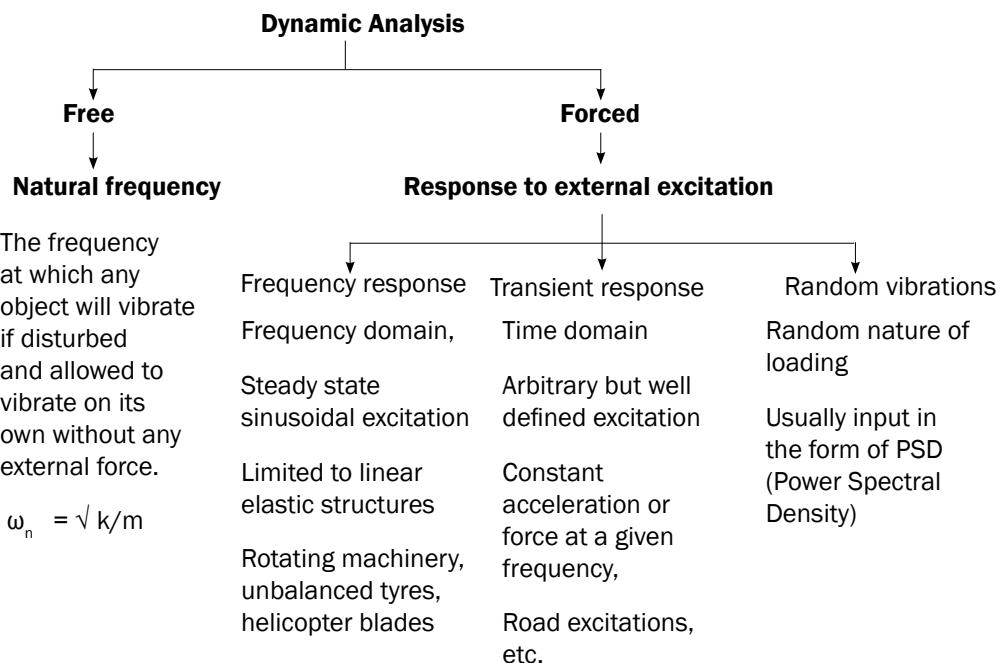
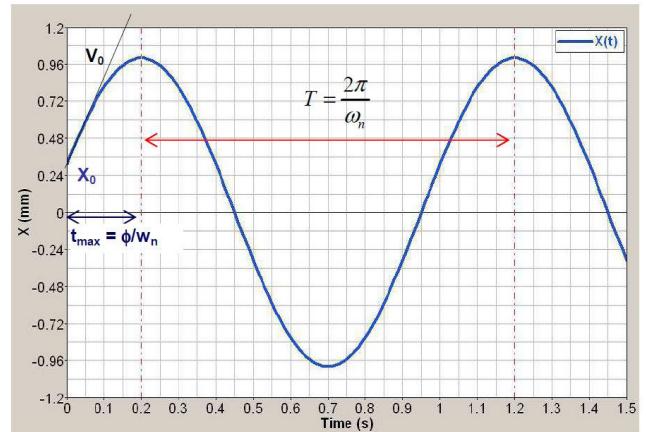
- The loads are applied as a function of time. $F(t)$
- The responses are time dependent. $X(t)$



Example Dynamic System

Dynamic analysis for simple structures like the one described above can be carried out manually. In general it is possible to find an analytical response for it or by using analytical tools it is possible to determine the mathematical functions that can represent the system responses.

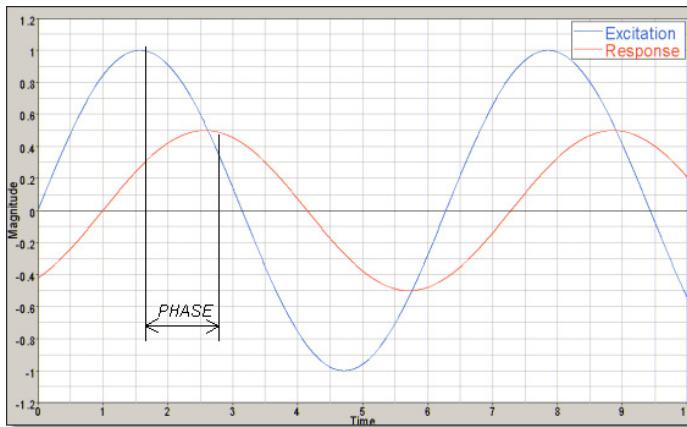
But for complex structures Finite Element Analysis (FEA) should be used to calculate the dynamic responses. This kind of analysis is well known as Structural Dynamic Analysis



Frequency Response Analysis

Frequency response analysis is used to calculate the response of a structure about steady state oscillatory excitation. Typical applications are noise, vibration and harshness (NVH) analysis of vehicles, rotating machinery, and transmissions.

Frequency response analysis is used to compute the response of the structure, which is actually transient, in a static frequency domain. The loading is sinusoidal. A simple case is a load of given amplitude at a specified frequency. The response occurs at the same frequency, and damping would lead to a phase shift.



Excitation and response of a frequency response analysis.

The loads can be forces, displacements, velocity, and acceleration. They are dependent on the excitation frequency Ω . The results from a frequency response analysis are displacements, velocities, accelerations, forces, stresses, and strains. The responses are usually complex numbers that are either given as magnitude and phase angle or as real and imaginary part.

Transient Response Analysis

Transient response analysis is used to calculate the response of a structure to time-dependent loads. Typical applications are structures subject to earthquakes, wind, explosions, or a vehicle going through a pothole. The loads are time-dependent forces and displacements. Initial conditions define the initial displacement and initial velocities in grid points.

The results of a transient response analysis are displacements, velocities, accelerations, forces, stresses, and strains. The responses are usually time-dependent.

The transient response analysis computes the structural responses solving the following equation of motion with initial conditions in matrix form.

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{B}\dot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{P}(t)$$

$$\mathbf{u}(t = 0) = \mathbf{u}_0$$

$$\dot{\mathbf{u}}(t = 0) = \mathbf{v}_0$$

The matrix K is the global stiffness matrix, the matrix M the mass matrix, and the matrix B is the damping matrix formed by the damping elements. The initial conditions are part of the problem formulation and are applicable for the direct transient response only. The equation of motion is integrated over time using the Newmark beta method. A time step and an end time need to be defined.

Response Spectrum Analysis

Response Spectrum Analysis (RSA) is a technique used to estimate the maximum response of a structure for a transient event. Maximum displacement, stresses, and/or forces may be determined in this manner. The technique combines response spectra for a prescribed dynamic loading with results of a normal modes analysis. The time-history of the responses is not available.

Response spectra describe the maximum response versus natural frequency of a 1-DOF system for a prescribed dynamic loading. They are employed to calculate the maximum modal response for each structural mode. These modal maxima may then be combined using various methods, such as the Absolute Sum (ABS) method or the Complete Quadratic Combination (CQC) method, to obtain an estimate of the peak structural response.

RSA is a simple and computationally inexpensive method to provide an approximation of peak response, compared to conventional transient analysis. The major computational effort is to obtain a sufficient number of normal modes in order to represent the entire frequency range of input excitation and resulting response. Response spectra are usually provided by design specifications; given these, peak responses under various dynamic excitations can be quickly calculated. Therefore, it is widely used as a design tool in areas such as seismic analysis of buildings.

While in a linear static analysis, the equation $F = K * u$ is solved, a dynamic analysis is based on some other equations:

$$[M] x'' + [C] x' + [K] x = F(t)$$

$x'' = d^2x / dt^2$ = acceleration, $x' = dx / dt$ = velocity, x = displacement

$[M] x' = 0$, $[C] x' = 0$, $[K]$ and $F(t) = \text{constant}$

- [Linear Static](#)

$[M] x'' = 0$, $[C] x' = 0$, $[K]$ is a function of $\{u\}$, $F(t) = \text{constant}$

- [Nonlinear Static](#)

$F(t) = 0$, $[C] x' = 0$ and $[M]$, $[K] = \text{constant}$

- [Free Vibration](#)

All the terms in above equation are present

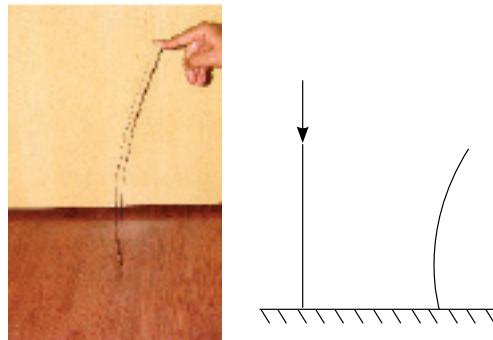
- [Forced Vibration](#)

Practical applications: Natural frequency is a characteristic and basic design property of any component, while forced vibrations is applicable for components subjected to force, displacement, velocity, or acceleration varying with respect to time or frequency.

4) Linear Buckling Analysis

Some key aspects:

- Applicable for only compressive load
- Slender beams and sheet metal parts
- Bending stiffness <<< Axial stiffness
- Large lateral deformation



The problem of linear buckling in finite element analysis is solved by first applying a reference level of loading, P_{Ref} , to the structure. A standard linear static analysis is then carried out to obtain stresses which are needed to form the geometric stiffness matrix K_G . The buckling loads are then calculated by solving an eigenvalue problem:

$$[K - \lambda K_G]x = 0$$

Where K is the stiffness matrix of the structure and λ is the multiplier to the reference load. The solution of the eigenvalue problem generally yields n eigenvalues λ , where n is the number of degrees of freedom (in practice, only a subset of eigenvalues is usually calculated). The vector x is the eigenvector corresponding to the eigenvalue.

The eigenvalue problem is solved using a matrix method called the Lanczos method. Not all eigenvalues are required. Only a small number of the lowest eigenvalues are normally calculated for buckling analysis.

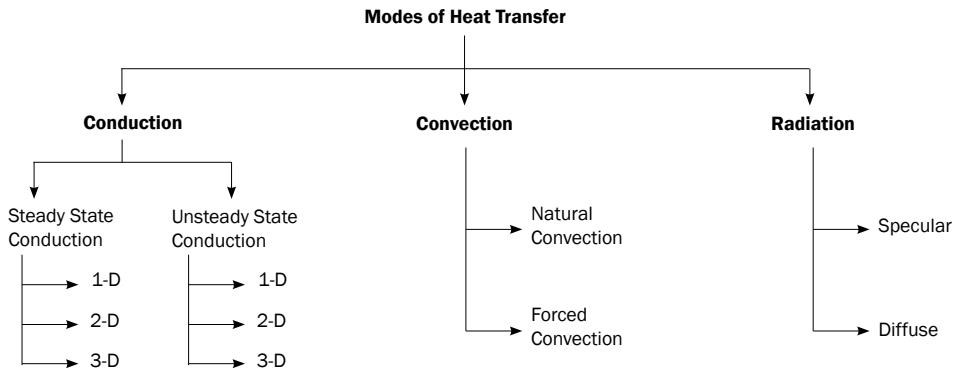
The lowest eigenvalue λ_{crit} is associated with buckling. The critical or buckling load is:

$$P_{\text{crit}} = \lambda_{\text{crit}} P_{\text{Ref}}$$

Output from software : Critical value of load.

Practical applications: Commonly used for civil engineering applications. Mechanical engineering applications - vacuum vessel, long gear shifter rod analysis, etc.

5) Thermal Analysis



Practical applications: Engine, radiator, exhaust system, heat exchangers, power plants, satellite design etc.

Linear Steady-State Heat Transfer Analysis

Heat transfer analysis solves for unknown temperatures and fluxes under thermal loading. Temperature represents the amount of thermal energy available, and fluxes represent the flow of thermal energy. Conduction deals with thermal energy exchange by molecular motion. Free convection deals with thermal energy exchange between solids and surrounding fluids. Thermal loading is defined as energy flows into and out of the system.

In linear steady state analysis, material properties such as conductivity and convection coefficient are linear. Temperature and fluxes at the final thermal equilibrium state are of interest. The basic finite element equation is:

$$([K_c] + [H])\{T\} = \{p\} \quad (1)$$

Where $[K_c]$ is the conductivity matrix, $[H]$ is the boundary convection matrix due to free convection, $\{T\}$ is an unknown nodal temperature, $\{p\}$ is the thermal loading vector. The system of linear equation is solved to find nodal temperature $\{T\}$.

Thermal load vector can be expressed as:

$$\{p\} = \{PB\} + \{PH\} + \{PQ\} \quad (2)$$

Where $\{PB\}$ is the power due to heat flux at boundary, $\{PH\}$ is the boundary convection vector due to convection and $\{PQ\}$ is the power vector due to internal heat generation. The matrix on the left hand side of equation (1) is singular unless temperature boundary conditions are specified. The equilibrium equation is solved simultaneously for the unknown temperatures using a Gauss elimination method that exploits the sparseness and symmetry for computational efficiency. Once the unknown temperatures at the nodal points of the elements are calculated, temperature gradient $\{\nabla T\}$ can be calculated according to element shape functions. Element fluxes can be calculated by using:

$$\{f\} = [k]\{\nabla T\} \quad (3)$$

Where $[k]$ is the conductivity of the material.

An analogy of heat transfer analysis and structural analysis is shown in the table below.

	Heat Transfer	Structural
Unknown	Temperature	Displacement
	Temperature gradient	Strain
	Flux	Stress
[Kc]	Conductivity matrix	Stiffness matrix
[H]	Boundary convection matrix	Elastic foundation stiffness matrix
{p}	Heat flux vector	Load vector
{PQP}	Element volumetric	Gravity load

6) Fatigue Analysis

In materials science, fatigue is the progressive and localized structural damage that occurs when a material is subjected to cyclic loading. The nominal maximum stress values are less than the ultimate tensile stress limit, and may be below the yield stress limit of the material.

Fatigue occurs when a material is subjected to repeated loading and unloading. If the loads are above a certain threshold, microscopic cracks will begin to form at the surface. Eventually a crack will reach a critical size, and the structure will suddenly fracture. The shape of the structure will significantly affect the fatigue life; square holes or sharp corners will lead to elevated local stresses where fatigue cracks can initiate. Round holes and smooth transitions or fillets are therefore important to increase the fatigue strength of the structure ([http://en.wikipedia.org/wiki/Fatigue_\(material\)](http://en.wikipedia.org/wiki/Fatigue_(material)))

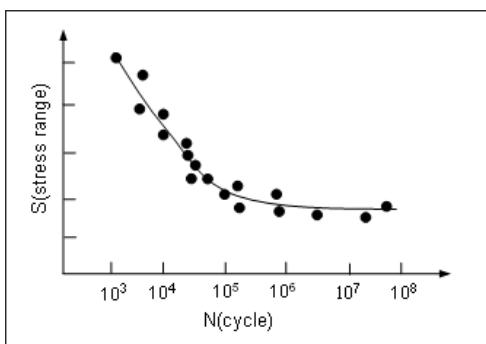
Fatigue analysis, using S-N (stress-life) and E-N (strain-life) approaches for predicting the life (number of loading cycles) of a structure under cyclical loading may be performed by using OptiStruct.

The stress-life method works well in predicting fatigue life when the stress level in the structure falls mostly in the elastic range. Under such cyclical loading conditions, the structure typically can withstand a large number of loading cycles; this is known as high-cycle fatigue. When the cyclical strains extend into plastic strain range, the fatigue endurance of the structure typically decreases significantly; this is characterized as low-cycle fatigue. The generally accepted transition point between high-cycle and low-cycle fatigue is around 10,000 loading cycles. For low-cycle fatigue prediction, the strain-life (E-N) method is applied, with plastic strains being considered as an important factor in the damage calculation.

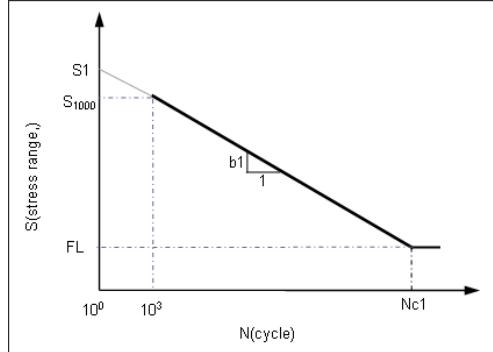
The Stress-Life (S-N) Approach

S-N Curve

The S-N curve, first developed by Wöhler, defines a relationship between stress and number of cycles to failure. Typically, the S-N curve (and other fatigue properties) of a material is obtained from experiment; through fully reversed rotating bending tests. Due to the large amount of scatter that usually accompanies test results, statistical characterization of the data should also be provided (certainty of survival is used to modify the S-N curve according to the standard error of the curve and a higher reliability level requires a larger certainty of survival).



When S-N testing data is presented in a log-log plot of alternating nominal stress amplitude S_a or range S_R versus cycles to failure N , the relationship between S and N can be described by straight line segments. Normally, a one or two segment idealization is used.



One segment S-N curves in log-log scale

$$S = S_1 (N_f)^{b_1} \quad \text{for segment 1} \quad (1)$$

where,

S is the nominal stress range

N_f are the fatigue cycles to failure

b_1 is the first fatigue strength exponent

S_1 is the fatigue strength coefficient

The S-N approach is based on elastic cyclic loading, inferring that the S-N curve should be confined, on the life axis, to numbers greater than 1000 cycles. This ensures that no significant plasticity is occurring. This is commonly referred to as high-cycle fatigue.

Damage Model

Palmgren-Miner's linear damage summation rule is used. Failure is predicted when:

$$\sum D_i = \sum \frac{n_i}{N_{if}} \geq 1.0 \quad (2)$$

where,

N_{if} is the material's fatigue life (number of cycles to failure) from its S-N curve at a combination of stress amplitude and mean stress level i ,

n_i is the number of stress cycles at load level i ,

D_i is the cumulative damage under n_i load cycle

The linear damage summation rule does not take into account the effect of the load sequence on the accumulation of damage due to cyclic fatigue loading. However, it has been proved to work well for many applications.

Cycle Counting

Cycle counting is used to extract discrete simple "equivalent" constant amplitude cycles from a random loading sequence. One way to understand "cycle counting" is as a changing stress-strain versus time signal. Cycle counting will count the number of stress-strain hysteresis loops and keep track of their range/mean or maximum/minimum values.

Rainflow cycle counting is the most widely used cycle counting method. It requires that the stress time history be rearranged so that it contains only the peaks and valleys and it starts either with the highest peak or the lowest valley (whichever is greater in absolute magnitude). Then, three consecutive stress points (S_1 , S_2 , and S_3) will define two consecutive ranges as $\Delta S_1 = |S_1 - S_2|$ and $\Delta S_2 = |S_2 - S_3|$. A cycle from S_1 to S_2 is only extracted if $\Delta S_1 \leq \Delta S_2$. Once a cycle is extracted, the two points forming the cycle

are discarded and the remaining points are connected to each other. This procedure is repeated until the remaining data points are exhausted.

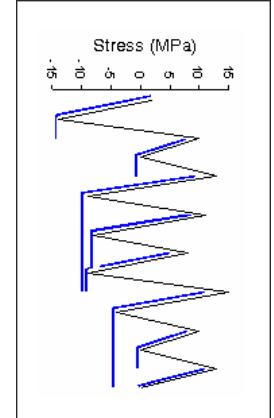


Figure: Determine cycles using rainflow cycle counting method

Equivalent Nominal Stress

Since S-N theory deals with uniaxial stress, the stress components need to be resolved into one combined value for each calculation point, at each time step, and then used as equivalent nominal stress applied on the S-N curve.

Various stress combination types are available with the default being “Absolute maximum principle stress”. In general “Absolute maximum principal stress” is recommended for brittle materials, while “Signed von Mises stress” is recommended for ductile material. The sign on the signed parameters is taken from the sign of the Maximum Absolute Principal value.

Mean Stress Influence

Generally S-N curves are obtained from standard experiments with fully reversed cyclic loading. However, the real fatigue loading could not be fully reversed and the normal mean stresses have significant effect on fatigue performance of components. Tensile normal mean stresses are detrimental and compressive normal mean stresses are beneficial, in terms of fatigue strength. Mean stress correction is used to take into account the effect of non-zero mean stresses.

The Gerber parabola and the Goodman line in Haigh's coordinates are widely used when considering mean stress influence, and can be expressed as:

$$\text{Gerber: } S_e = S_a / (1 - (S_m / S_u)^2) \quad (3)$$

$$\text{Goodman: } S_e = S_a / (1 - S_m / S_u) \quad (4)$$

where,

$$\text{Mean stress } S_m = (S_{\max} + S_{\min}) / 2,$$

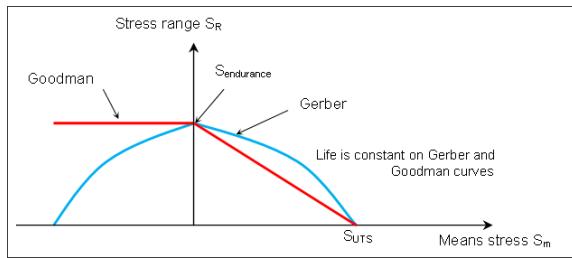
$$\text{Stress amplitude } S_a = (S_{\max} - S_{\min}) / 2,$$

S_e is the stress range for fully reversed loading that is equivalent to the load case with a stress range S_R and a mean stress S_m ,

S_u is ultimate strength.

The Gerber method treats positive and negative mean stress correction in the same way that mean stress always accelerates fatigue failure, while the Goodman method ignores the negative mean stress. Both methods give conservative result for compressive mean stress. The Goodman method is recommended for brittle material while the Gerber method is recommended for ductile material. For the Goodman method, if the tensile mean stress is greater than UTS, the damage will be greater than 1.0. For Gerber method, if the mean stress is greater than UTS, no matter tensile or compressive, the damage will be greater than 1.0.

A Haigh diagram characterizes different combinations of stress amplitude and mean stress for a given number of cycles to failure.



Haigh diagram and mean stress correction methods

The Strain-Life (E-N) Approach

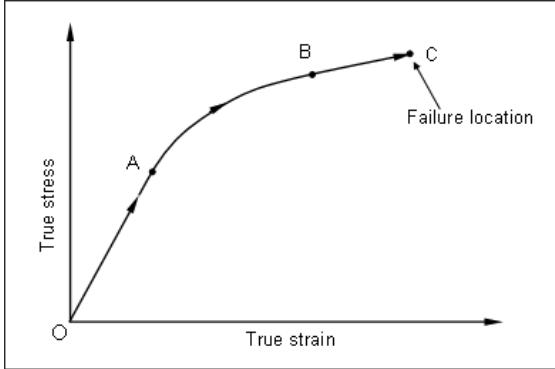
Monotonic Stress-Strain Behavior

Relative to the current configuration, the true stress and strain relationship can be defined as:

$$\sigma = P / A \quad (5)$$

$$\varepsilon = \int_{l_0}^l \frac{dl}{l} = \ln \left(1 + \frac{l - l_0}{l_0} \right) \quad (6)$$

Where A is the current cross-section area, l is the current specimen length, l_0 is the initial specimen length, σ and ε are the true stress and strain, respectively. The figure below depicts the monotonic stress-strain curve in true stress-strain space. In the whole process, the stress continues increasing to a large value until the specimen fails at C.



Monotonic stress-strain curve

The curve in the above figure is comprised of two typical segments, namely the elastic segment OA and plastic segment AC. The segment OA keeps the linear relationship between stress and elastic strain following Hooke Law:

$$\sigma = E \varepsilon_{el} \quad (7)$$

Where E is elastic modulus, ε_{el} is elastic strain. The formula can also be rewritten as:

$$\varepsilon_{el} = \sigma / E \quad (8)$$

by expressing elastic strain in terms of stress. For most of materials, the relationship between the plastic strain and the stress can be represented by a simple power law of the form:

$$\sigma = K (\varepsilon_{pl})^n \quad (9)$$

Where ε_{pl} is plastic strain, K is strength coefficient, n is work hardening coefficient. Similarly, the plastic strain can be expressed

in terms of stress as:

$$\varepsilon_{pl} = (\sigma / K)^{1/n} \quad (10)$$

The total strain induced by loading the specimen up to point B or D is the sum of plastic strain and elastic strain:

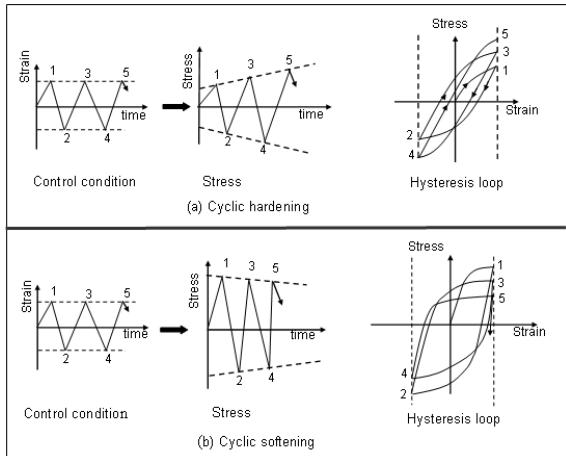
$$e = \varepsilon_{el} + \varepsilon_{pl} = \sigma / E + (\sigma / K)^{1/n} \quad (11)$$

Cyclic Stress-Strain Curve

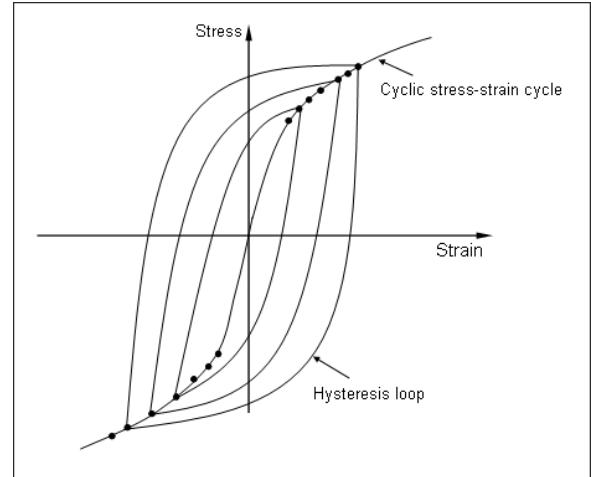
Material exhibits different behavior under cyclic load compared with that of monotonic load. Generally, there are four kinds of response.

- stable state
- cyclically hardening
- cyclically softening
- softening or hardening depending on strain range

Which response will occur depends on its nature and initial condition of heat treatment. The image below illustrates the effect of cyclic hardening and cyclic softening where the first two hysteresis loops of two different materials are plotted. In both cases, the strain is constrained to change in fixed range, while the stress is allowed to change arbitrarily. If the stress range increases relative to the former cycle under fixed strain range, as shown in the upper part of the image, it is called cyclic hardening; otherwise, it is called cyclic softening, as shown in the lower part of the figure. Cyclic response of material can also be described by specifying the stress range and leaving strain unconstrained. If the strain range increases relative to the former cycle under fixed stress range, it is called cycle softening; otherwise, it is called cyclic hardening. In fact, the cyclic behavior of material will reach a steady state after a short time which generally occupies less than 10 percent of the material total life. Through specifying different strain ranges, a series of hysteresis loops at steady state can be obtained.



Material cyclic response (a) Cyclic hardening; (b) Cyclic softening



Definition of stable stress-strain curve

By placing these hysteresis loops in one coordinate system, as shown below, the line connecting all the vertices of these hysteresis loops determine cyclic stress-strain curve which can be expressed in the similar form with monotonic stress-strain curve as:

$$\varepsilon = \varepsilon_{el} + \varepsilon_{pl} = \sigma / E + (\sigma / K')^{1/n'} \quad (12)$$

where, K' is cyclic strength coefficient, n' is strain cyclic hardening exponent.

Hysteresis Loop Shape

Bauschinger observed that after the initial load had caused plastic strain, load reversal caused materials to exhibit anisotropic behavior. Based on experimental evidence, Massing put forward the hypothesis that a stress-strain hysteresis loop is geometrically similar to the cyclic stress strain curve, but with twice the magnitude. This implies that when the quantity $(\Delta\varepsilon, \Delta\sigma)$ is two times of (ε, σ) the stress-strain cycle will lie on the hysteresis loop. This can be expressed with formulas:

$$\Delta\sigma = 2\sigma \quad (13)$$

$$\Delta\varepsilon = 2\varepsilon \quad (14)$$

Expressing σ in terms of $\Delta\sigma$, ε in terms of $\Delta\varepsilon$, and substituting it into Eq. 12, the hysteresis loop formula can be deduced as:

$$\Delta\varepsilon = \Delta\sigma / E + 2 (\Delta\sigma / K')^{1/n} \quad (15)$$

Strain-Life Approach

Almost a century ago, Basquin observed the linear relationship between stress and fatigue life in log scale when the stress is limited. He put forward the following fatigue formula controlled by stress:

$$\sigma_a = \sigma_f' (2N_f)^b \quad (16)$$

where σ_a is stress amplitude, σ_f' fatigue strength coefficient, b fatigue strength exponent. Later in the 1950s, Coffin and Manson independently proposed that plastic strain may also be related with fatigue life by a simple power law:

$$\varepsilon_{pa} = \varepsilon_f' (2N_f)^c \quad (17)$$

where ε_{pl} is plastic strain amplitude, ε_f' fatigue ductility coefficient, c fatigue ductility exponent. Morrow combined the work of Basquin, Coffin and Manson to consider both elastic strain and plastic strain contribution to the fatigue life. He found out that the total strain has more direct correlation with fatigue life. By applying Hooke Law, Basquin rule can be rewritten as:

$$\varepsilon_{ea} = \frac{\sigma_a}{E} = \frac{\sigma_f'}{E} (2N_f)^b \quad (18)$$

where ε_{ea} is elastic strain amplitude. Total strain amplitude, which is the sum of the elastic strain and plastic stain, therefore, can be described by applying Basquin formula and Coffin-Manson formula:

$$\varepsilon_a = \varepsilon_{ea} + \varepsilon_{pa} = \frac{\sigma_f'}{E} (2N_f)^b + \varepsilon_f' (2N_f)^c \quad (19)$$

where ε_a is the total strain amplitude, the other variable is the same with above. The image below illustrates three methods in log scale in stress-life space. Two straight lines, which represent Basquin formula and Coffin-Manson rule respectively, intersect at a point where elastic strain is equal to the plastic strain and the fatigue life predicted by the two methods is the same. The fatigue life at the intersection point is called transition life and can be calculated as:

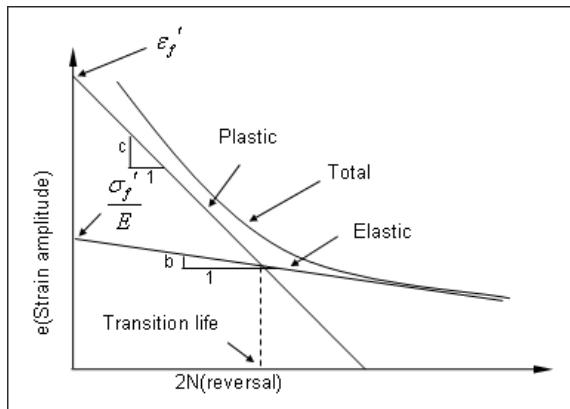
$$2N_t = \left(\varepsilon_f' E / \sigma_f' \right)^{1/(b-c)} \quad (20)$$

by combining Eq.17 and Eq.18, at the same time, applying the conditions:

$$\varepsilon_{ea} = \varepsilon_{pa} \quad (21)$$

$$N_t = N_f \quad (23)$$

where N_t is transition life. When fatigue life is less than the transition life, plastic strain plays the controlling role in life prediction; otherwise, elastic strain plays the key role.



Strain-life curve in log scale

Damage Accumulation Model

In the E-N approach, we use the same damage accumulation model as the S-N approach, which is Palmgren-Miner's linear damage summation rule.

Mean Stress Influence

The fatigue experiments carried out in the laboratory are always fully reversed, whereas in practice, the mean stress is inevitable, thus the fatigue law established by the fully reversed experiments must be corrected before applied to engineering problems. Morrow is the first to consider the effect of mean stress through introducing the mean stress σ_0 in fatigue strength coefficient by:

$$\varepsilon_{ea} = \frac{(\sigma_f' - \sigma_0)}{E} (2N_f)^b \quad (23)$$

Thus the entire fatigue life formula becomes

$$\varepsilon_a = \frac{(\sigma_f' - \sigma_0)}{E} (2N_f)^b + \varepsilon_f' (2N_f)^c \quad (24)$$

Morrow's equation is consistent with the observation that mean stress effects are significant at low value of plastic strain and of little effect at high plastic strain.

Smith, Watson and Topper proposed a different method to account for the effect of mean stress by considering the maximum stress during one cycle (for convenience, this method is called SWT in the following). In this case, the damage parameter is modified as the product of the maximum stress and strain range in one cycle. For a fully reversed cycle, the maximum stress is given by:

$$\sigma_{max} = \sigma_f' (2N_f)^b \quad (25)$$

By multiplying Eq.19 with Eq.25, it can be rewritten as:

$$\sigma_{max} \varepsilon_a = \frac{\sigma_f'}{E} (2N_f)^{2b} + \sigma_f' \varepsilon_f' (2N_f)^{b+c} \quad (26)$$

The SWT method will predict that no damage will occur when the maximum stress is zero or negative, which is not consistent with the reality.

When comparing the two methods, the SWT method predicted conservative life for loads predominantly tensile, whereas, the Morrow approach provides more realistic results when the load is predominantly compressive.

Neuber Correction

Strain-life analysis is based on the fact that many critical locations such as notch roots have stress concentration, which will have obvious plastic deformation during the cyclic loading before fatigue failure. Thus, the elastic-plastic strain results are essential for performing strain-life analysis. Neuber correction is the most popular practice to correct elastic analysis results into elastic-plastic results.

In order to derive the local stress from the nominal stress that is easier to obtain, the concentration factors are introduced such as the local stress concentration factor K_{σ} and the local strain concentration factor K_{ε}

$$K_{\sigma} = \sigma / S \quad (27)$$

$$K_{\varepsilon} = \varepsilon / e \quad (28)$$

where σ is local stress, ε local strain, S nominal stress, e nominal strain. If nominal stress and local stress are both elastic, the local stress concentration factor is equal to the local strain concentration factor.

However, if the plastic strain is present, the relationship between K_{σ} and K_{ε} no longer holds. Thereafter, focusing on this situation, Neuber introduced a theoretically elastic stress concentration factor K_t defined as:

$$K_t^2 = K_{\sigma} K_{\varepsilon} \quad (29)$$

Substitute Eq.27 and Eq.28 into Eq.29, the theoretical stress concentration factor K_t can be rewritten as:

$$K_t^2 = \left(\frac{\sigma}{S} \right) \left(\frac{\varepsilon}{e} \right) \quad (30)$$

Through linear static FEA, the local stress instead of nominal stress is provided, which implies the effect of the geometry in Eq.30 is removed, thus we can set K_t as 1 and rewrite Eq.30 as:

$$\sigma \varepsilon = \sigma_e \varepsilon_e \quad (31)$$

where σ_e , ε_e is locally elastic stress and locally elastic strain obtained from elastic analysis, σ , ε the stress and strain at the presence of plastic strain. Both σ and ε can be calculated from Eq.31 together with the equations for the cyclic stress-strain curve and hysteresis loop.

Other Factors Affecting Fatigue

- **Surface Condition** (Finish and Treatment)

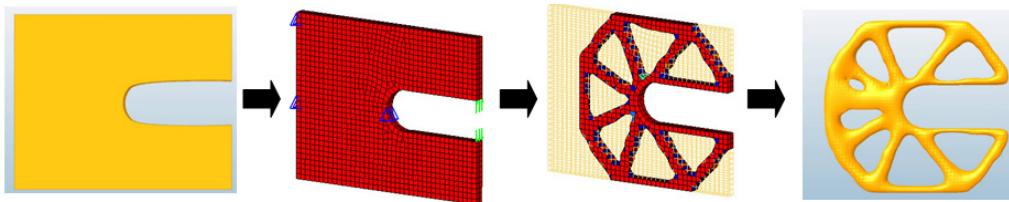
Surface condition is an extremely important factor influencing fatigue strength, as fatigue failures nucleate at the surface. Surface finish and treatment factors are considered to correct the fatigue analysis results. Surface treatment can improve the fatigue strength of components.

- **Fatigue Strength Reduction Factor**

In addition to the factors mentioned above, there are various other factors that could affect the fatigue strength of a structure, e.g., notch effect, size effect, loading type. Fatigue strength reduction factor K_f is introduced to account for the combined effect of all such corrections.

Practical applications: Applicable to all the components subjected to dynamic loading (e.g. all automobile components). Fatigue accounts for 90 % of failure in the real world.

7) Optimization

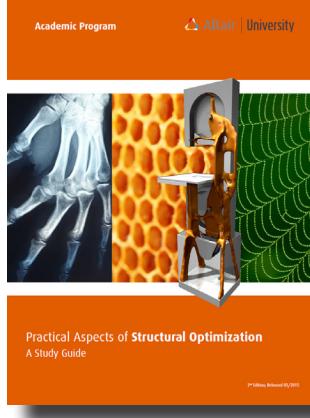


Optimization is clearly one of the overall strengths of Altair and HyperWorks. You may distinguish optimization methods with respect to its position in the design phase, i.e. concept design optimization such as topology, topography and free size optimization, and “fine tuning” optimization disciplines such as size or shape optimization. Alternatively, you can distinguish according to the design variable, i.e. which variable of the model is modified/ altered during the optimization. For instance, the design variable of a topology optimization is the density of elements, whereas in size optimization the thickness of a sheet metal may be varied.

Practical applications: Applicable to any component that is over or under designed.

A comprehensive description of “optimization” is also available at <http://en.wikipedia.org/wiki/Optimization> or you can view the HyperWorks help for Optimization.

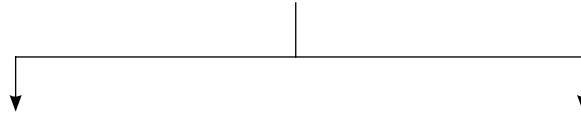
You may also find the free study guide entitled “Practical Aspects Of Structural Optimization” available on the Academic Website helpful.



8) Computational Fluid Dynamics (CFD)

A fluid is a substance that continuously deforms under an applied shear stress regardless of the magnitude of the applied stress. Gas and liquids are both fluids. Fluid mechanics deals with the study of fluid, its properties, and its behavior.

Fluid Mechanics



Fluid Statics (Hydrostatics)

- Fluid at rest
- Easy from a calculation point of view
- No variation with respect to time, less realistic results
- Linear equations
- Civil engineering: Dam design

Fluid Dynamics

- Fluid in motion
- Calculations are more complex
- Variation with respect to time considered
- Nonlinear equations
- Aerospace, automobile, marine and valves

Computational Fluid Dynamics (CFD) is the branch of fluid mechanics where numerical methods are used to analyze fluid dynamics problems. It is based on the Navier –Stokes equations (mass, momentum, and energy conservation equilibrium equations).

Flow Classification				
Viscosity	Dimensions	Turbulent/ Laminar	Compressibility	Velocity
Inviscid: no viscosity	1D: e.g. flow through a nozzle	<ul style="list-style-type: none"> Laminar: Reynolds's number < 2300. Flow through a water tap at low velocities 	Incompressible: high density fluids flowing at sub sonic velocity (mach number<1) e.g. water, oil	Sub Sonic $M < 1.0$
Viscid: Viscous flow. Most common	2D: e.g. flow through a pipe or over an airfoil	<ul style="list-style-type: none"> Turbulent: Reynold's number > 2300. Flow through a water tap at higher velocities 	Compressible: low density fluids flowing at sonic or supersonic velocity e.g. discontinuities like shock and expansion waves	Trans sonic $0.85 < M < 1.20$ Represents weak shocks
	3D: e.g. flow through a valve, flow over an automobile, aircraft, or missile			Super sonic $1.20 < M < 4.0$ Hypersonic $M > 6.0$ e.g. Space shuttle entering earth's atmosphere.

Practical applications: Drag prediction and stream lining of a car, combustion chamber design to check an optimum fuel - air mixing, aeroplane design, etc.

9) Crash Analysis

Crash Analysis		
Structural Crashworthiness Or Full Dynamic / Impact Simulations	Drop Test Simulations	Occupant Safety
To find deformation, stress, and energy absorbing capacity of various structural components of a vehicle hitting a stationary or moving object. The component is said to be crashworthy (safe), if it meets the plastic strain and energy targets . Applications: Frontal, side, rear, roof crush, car hitting a pole / wall etc.	Drop test is a free fall test carried out to check the structural integrity of a component. Applications: Black box of an aircraft, mobile phone, consumer goods such as TV, fridge, etc.	To find the effects of crash on the human body and making the ride safe for the driver as well as the passengers . Several regulations exist in different countries to ensure a proper certification, e.g.: FMVSS (Federal Motor Vehicle Safety Standards) in the USA, ECE (Economic Commission of Europe) regulation in Europe. In India, the ARAI has set up standard procedures for the automobile industry and called AIS (Automotive Industry Standards)

10) Noise, Vibration And Harshness, NVH

Sound Radiation Or Scattering Or Uncoupled Problems	Coupled Or Vibroacoustics Problems
<p>This predicts how much sound pressure level is felt by a vibrating source at a certain distance as a function of the solid angle. A typical example is how much sound level is felt due to a horn or a silencer vibrating at a certain distance.</p> <p>These problems are solved by Boundary Element Method.</p>	<p>This is when there is a clear interaction of a structure and fluid cavity. A typical example is when there is a noise level felt at the driver's right ear due to the engine vibration in an idle condition.</p> <p>These problems are solved by Finite Element Method.</p>

NVH problems are broadly classified into structure born and air born sound.

Practical applications: Computing the sound pressure level is of utmost importance to automobile, airplane, and aerospace designers as customers always prefer a low noise level. Computing the response at the driver's feet (brake pedal), mirror mounts, steering column, and seats plays a crucial role as the driver must be comfortable. Also predicting the sound quality and radiation at a certain distance from the car is important.

3 Introduction To Finite Element Analysis

This chapter includes material from the book “**Practical Finite Element Analysis**” as well as information from the HyperWorks Help Documentation. Additional material was added by Matthias Goelke and Jan Grasmannsdorf.

3.1 A Brief Summary

As mentioned in the introduction to this guide we deliberately refrained from including a chapter about the theory of the Finite Element Method. The reason is rather simple - the book shelves in the libraries are filled up with thousands of books about FEM already. In addition, in the web you will find a countless number of online learning material like reports, FEM lecture notes, videos and so on.

Hence, the paragraph below is meant to give FEM beginners a “feeling” on what is happening in the background.

Methods to Solve Any Engineering Problem		
Analytical Method	Numerical Method	Experimental Method
<ul style="list-style-type: none">• Classical approach• 100% accurate results• Closed form solution• Applicable only for simple problems like cantilever and simply supported beams, etc.• Complete in itself <p>Though analytical methods could also give approximate results if the solution is not closed form, in general analytical methods are considered as closed form solutions i.e. 100% accurate.</p>	<ul style="list-style-type: none">• Mathematical representation• Approximate, assumptions made• Applicable even if a physical prototype is not available (initial design phase)• Real life complex problems• Results cannot be believed blindly. Certain results must be validated by experiments and/or analytical method. <p>Finite Element Method: Linear, nonlinear, buckling, thermal, dynamic, and fatigue analysis Boundary Element Method: Acoustics, NVH Finite Volume Method: CFD (Computational Fluid Dynamics) and Computational Electromagnetics Finite Difference Method: Thermal and Fluid flow analysis (in combination with FVM)</p>	<ul style="list-style-type: none">• Actual measurement• Time consuming and needs expensive set up• Applicable only if physical prototype is available• Results cannot be believed blindly and a minimum of 3 to 5 prototypes must be tested <p>• Strain gauge • Photo elasticity • Vibration measurements • Sensors for temperature and pressure, etc. • Fatigue test</p>

Procedure For Solving Any Analytical Or Numerical Problem

There are two steps to solving analytical or numerical problems:

Step 1) Writing of the governing equation – Problem definition, or in other words, formulating the problem in the form of a mathematical equation.

Step 2) Mathematical solution of the governing equation.

The final result is the summation of step 1 and step 2. The result will be 100% accurate when there is no approximation at either of the steps (analytical method).

Numerical methods make an approximation at step 1 and at step 2, therefore all numerical methods are approximate.

	Analytical "Approximation"	Numerical "Approximation"
Step 1	Little or no approximation	✓
Step 2	Little or no approximation	✓
Accuracy	High accuracy	Approximate results

Brief Introduction To Different Numerical Methods

1) Finite Element Method (FEM):

FEM is the most popular numerical method.

The Finite Element Method (FEM) is a numerical technique used to determine the approximated solution for a partial differential equations (PDE) on a defined domain (W). To solve the PDE, the primary challenge is to create a function base that can approximate the solution. There are many ways of building the approximation base and how this is done is determined by the formulation selected. The Finite Element Method has a very good performance to solve partial differential equations over complex domains that can vary with time.

Applications - Linear, nonlinear, buckling, thermal, dynamic and fatigue analysis. FEM will be discussed later.

Are FEA and FEM different?

Finite Element Method (FEM) and Finite Element Analysis (FEA) are one and the same. The term "FEA" is more popular in industries while "FEM" is more popular at universities.

Many times there is confusion between FEA, FEM, and one more similar but different term FMEA (Failure Mode Effect Analysis). FEA/FEM is used by design or Research and Development departments only, while FMEA is applicable to all of the departments.

2) Boundary Element Method (BEM):

This is a very powerful and efficient technique to solve acoustics or NVH problems. Just like the finite element method, it also requires nodes and elements, but as the name suggests it only considers the outer boundary of the domain. So when the problem is of a volume, only the outer surfaces are considered. If the domain is of an area, then only the outer periphery is considered. This way it reduces the dimensionality of the problem by a degree of one and thus solving the problem faster.

The Boundary Element Method (BEM) is a numerical method of solving linear PDE which have been formulated as integral equations. The integral equation may be regarded as an exact solution of the governing partial differential equation. The BEM attempts to use the given boundary conditions to fit boundary values into the integral equation, rather than values throughout the space defined by a partial differential equation. Once this is done, in the post-processing stage, the integral equation can then be used again to calculate numerically the solution directly at any desired point in the interior of the solution domain. The boundary element method is often more efficient than other methods, including finite elements, in terms of computational resources for problems where there is a small surface/volume ratio. Conceptually, it works by constructing a "mesh" over the modeled surface. However, for many problems boundary element methods are significantly less efficient than volume-discretization methods like FDM, FVM or FEM.

3) Finite Volume Method (FVM):

The Finite Volume Method (FVM) is a method for representing and evaluating partial differential equations as algebraic equations [LeVeque, 2002; Toro, 1999]. It is very similar to FDM, where the values are calculated at discrete volumes on a generic geometry. In the FVM, volume integrals in a partial differential equation that contain a divergence term are converted to surface integrals, using the divergence theorem. These terms are then evaluated as fluxes at the surfaces of each finite volume. Because the flux entering a given volume is identical to that leaving the adjacent volume, these methods are conservative. Another advantage of the finite volume method is that it is easily formulated to allow for unstructured meshes. The method is used in many computational fluid dynamics packages.

4) Finite Difference Method (FDM):

Finite Element and Finite Difference Methods share many common things. In general the Finite Difference Method is described as a way to solve differential equation. It uses Taylor's series to convert a differential equation to an algebraic equation. In the conversion process, higher order terms are neglected. It is used in combination with BEM or FVM to solve thermal and CFD coupled problems.

Finite Difference Method is the discretization of partial differential equations while Finite Element Method, Boundary Element Method and Finite Volume Method are the discretization of the integral form of the equations

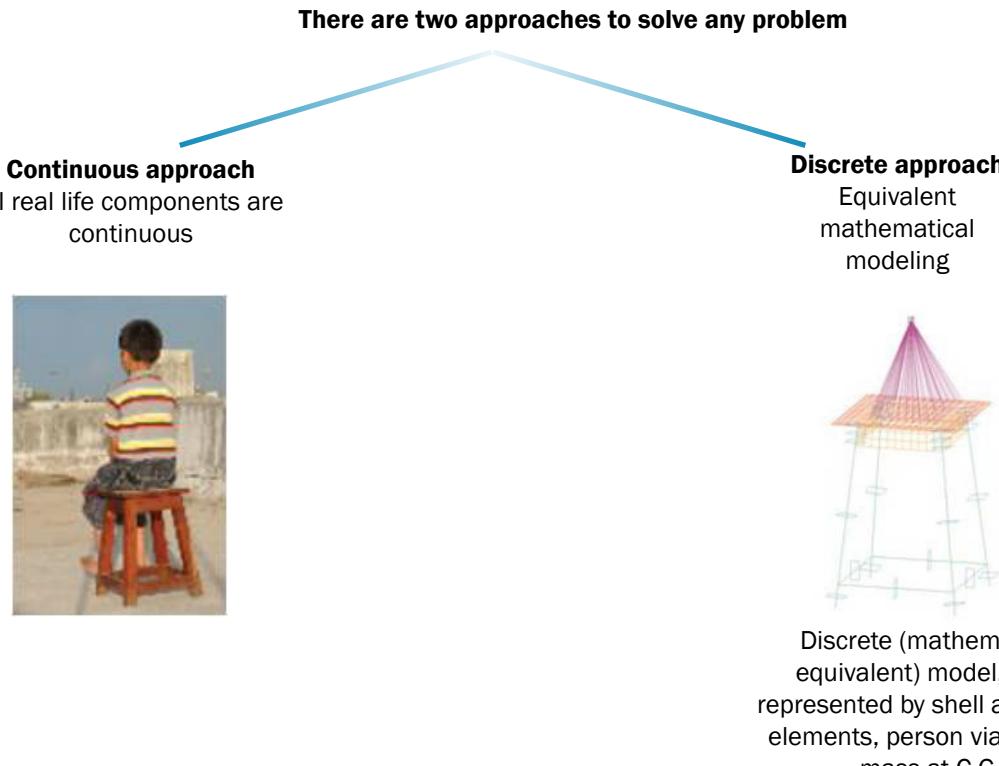
Question: Is it possible to use all of the methods listed above (FEA, BEA, FVM, FDM) to solve the same problem (for example, a cantilever problem)?

The answer is YES! But the difference is in the accuracy achieved, programming ease, and the time required to obtain the solution.

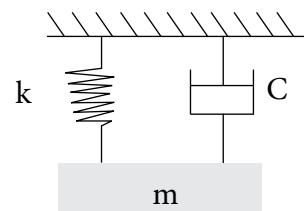
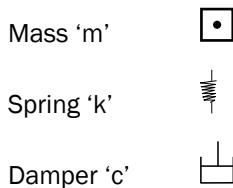
When internal details are required (such as stresses inside the 3D object) BEM will lead to poor results (as it only considers the outer boundary), while FEM, FDM, or FVM are preferable. FVM has been used for solving stress problems but it is well suited for computational fluid dynamics problems where conservation and equilibrium is quite natural. FDM has limitations with complicated geometry, assembly of different material components, and the combination of various types of elements (1D, 2D and 3D). For these types of problems FEM is far ahead of its competitors.

Discretization Of Problem:

All real life objects are continuous. This means there is no physical gap between any two consecutive particles. As per material science, any object is made up of small particles, particles of molecules, molecules of atoms, and so on and they are bonded together by the force of attraction. Solving a real life problem with the continuous material approach is difficult. The basis of all numerical methods is to simplify the problem by discretizing (discontinuation) it. In other words, nodes work like atoms and the gap in between the nodes is filled by an entity called an element. Calculations are made at the nodes and results are interpolated for the elements.



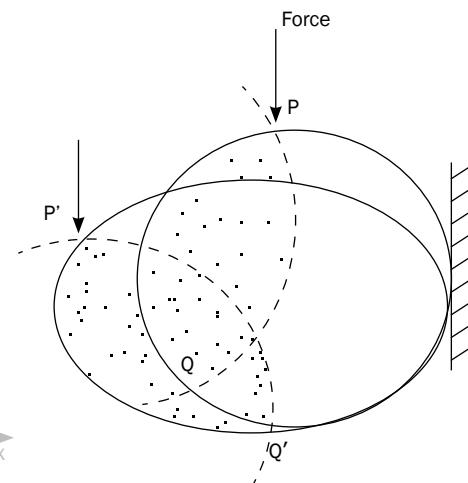
From a mechanical engineering point of view, any component or system can be represented by three basic elements:



All the numerical methods including the Finite Element Method follow the discrete approach. Meshing (nodes and elements) is nothing but the discretization of a continuous system with infinite degrees of freedom to a finite degrees of freedom.

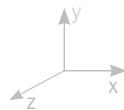
What Is DOF (Degree Of Freedom) ?

In this example, an object is fixed at one end and a force is applied at the point "P". Due to the force, the object deforms and point P gets shifted to new position P'.



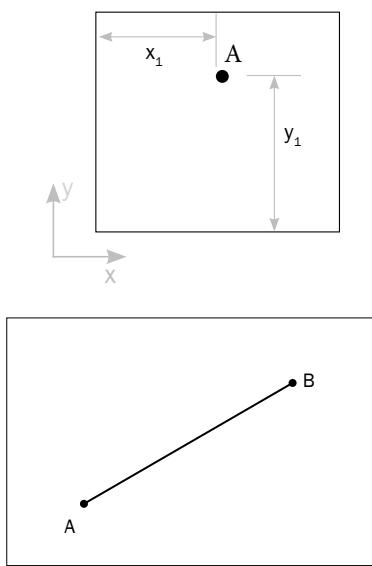
When Can We Say That We Know The Solution To The Above Problem?

If and only if we are able to define the deformed position of each and every particle completely.

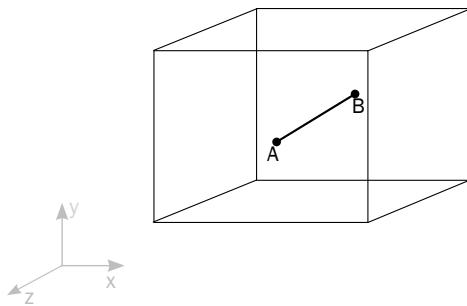


The minimum number of parameters (motion, coordinates, temperature, etc.) required to define the position and state of any entity completely in space is known as degrees of freedom (dof).

Consider the following 2-D (planar) problem. Suppose the origin is at the bottom left corner and is known. To define the position of point A completely with respect to the origin, we need two parameters i.e. x_1 and y_1 , in other words 2 dofs (translation x and y).



Consider that the point A is a part of a line, now one angle should also be defined in addition to the two translations i.e. 3 dofs (two translations and one rotation).

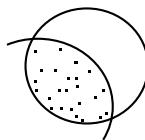


Suppose points A and B are shifted out of the plane and the line is rotated arbitrarily with respect to all of the three axes. The minimum number of parameters to define the position of point A completely would be 6 dofs i.e. 3 translations (U_x, U_y, U_z) and 3 rotations ($\theta_x, \theta_y, \theta_z$).

DOF is a very important concept. In FEA, we use it for the individual calculation points. **The total DOFs for a given mesh model is equal to the number of nodes multiplied by the number of dof per node.**

All of the elements do not always have 6 dofs per node. The number of dofs depends on the type of element (1D, 2D, 3D), the family of element (thin shell, plane stress, plane strain, membrane, etc.), and the type of analysis. For example, for a structural analysis, a thin shell element has 6 dof/node (displacement unknown, 3 translations and 3 rotations) while the same element when used for thermal analysis has single dof /node (temperature unknown).

For a new user, it is a bit confusing but there is a lot of logical, engineering, and mathematical thinking behind assigning the specific number of dofs to different element types and families.



No. of points = ∞
DOF per point = 6
Total equations = ∞



No. of nodes = 8
DOF per node = 6
Total equations = 48

Why Do We Carry Out Meshing? What Is FEM / FEA?

FEM

- A numerical method
- Mathematical representation of an actual problem
- Approximate method

The Finite Element Method only makes calculations at a limited (Finite) number of points and then interpolates the results for the entire domain (surface or volume).

Finite – Any continuous object has infinite degrees of freedom and it is not possible to solve the problem in this format. The Finite Element Method reduces the degrees of freedom from infinite to finite with the help of discretization or meshing (nodes and elements).

Element – All of the calculations are made at a limited number of points known as nodes. The entity joining nodes and forming a specific shape such as quadrilateral or triangular is known as an Element. To get the value of a variable (say displacement) anywhere in between the calculation points, an interpolation function (as per the shape of the element) is used.

Method - There are 3 methods to solve any engineering problem. Finite element analysis belongs to the numerical method category.

It is not necessary to remember the mathematical definition of FEM word by word as given in theoretical text books. Rather what is important is to understand the concept and then be able to describe it in your own words.

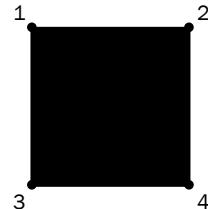
How The Results Are Interpolated From A Few Calculation Points

It is ok that FEA is making all the calculations at a limited number of points, but the question is how it calculates values of the unknown somewhere in between the calculation points.

This is achieved by interpolation. Consider a 4 noded quadrilateral element as shown in the figure below. A “quad4” element uses the following linear interpolation formula:

$$u = a_0 + a_1 x + a_2 y + a_3 xy$$

FEA calculates the values at the outer nodes 1, 2, 3, 4 i.e. a_0, a_1, a_2, a_3 are known.

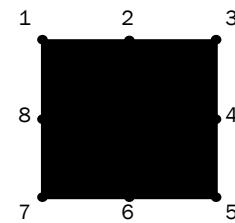


4 noded (linear) quad

The value of the variable anywhere in between could be easily determined just by specifying x and y coordinates in above equation.

For an 8 noded quadrilateral, the following parabolic interpolation function is used:

$$u = a_0 + a_1 x + a_2 y + a_3 xy + a_4 x^2 + a_5 y^2 + a_6 x^2 y + a_7 xy^2$$

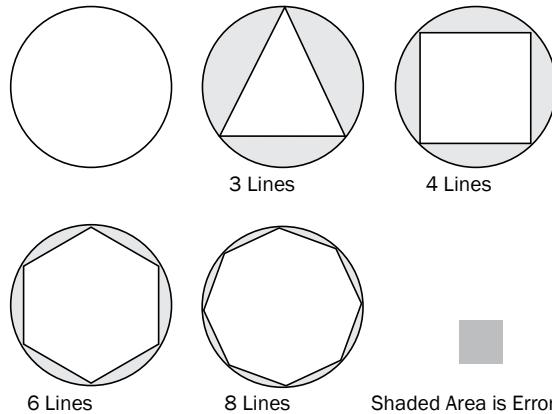


8 noded (parabolic) quad

How Is The Accuracy If We Increase The Number Of Calculation Points (Nodes And Elements)?

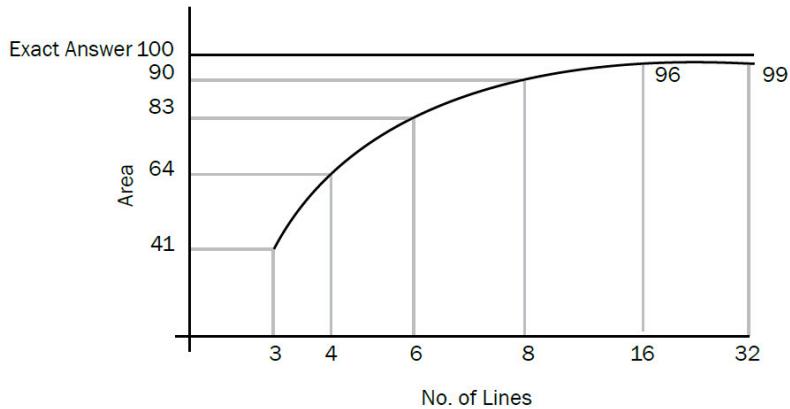
In general, increasing the number of calculation points improves the accuracy.

Suppose somebody gives you 3 straight lines and asks you to best fit it in a circle, then find the area of the triangle and compare it with the circle area. This is then repeated with 4, 6, 8, 16, 32 and 64 lines.



By increasing the number of lines, the error margin reduces. The number of straight lines is equivalent to the number of elements in Finite Element Analysis.

The exact answer for the area of the circle (πr^2) is 100. 3 lines gives the answer of 41, while 4 lines gives 64, and so on. An answer of 41 or 64 is not at all acceptable, but 80 or 90 is, considering the time spent and the relative design concept.



If a higher number of nodes and elements leads to a higher accuracy, then why not always create a very fine mesh with the maximum possible number of nodes and elements? The reason is because the solution time is directly proportional to $(dof)^n$.

n can be 1 to 4, depending on the type of analyses and solver. Also, large size models are not easy to handle on computers due to the graphics card memory limitations. The analyst has to maintain a fine balance between the desired level of accuracy and the element size (dof) that can be handled satisfactorily using the available hardware resources.

Assume the Analytical Method approach gives answers very close to 100 and the time it takes is 1 month, while the Finite Element Analysis with a reasonable mesh size gives an answer of 90 within 1 day. In industry, getting fast solutions with logical or reasonable accuracy is more important than absolute accuracy.

That's why the analytical method approach is also known as the Scientists way to solve any problem while the Numerical method is the Engineers way to solve the problem.

Advantages Of FEA

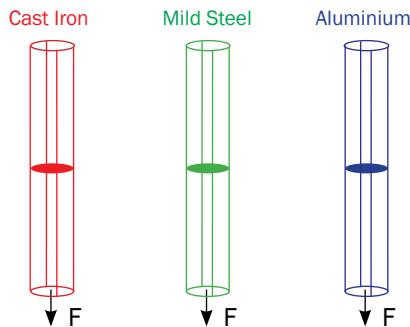
- Visualization ↑
- Design cycle time ↓
- No. of prototypes ↓
- Testing ↓
- Optimum design

Visualization of results: For simple geometries such as a simply supported beam or a cantilever beam, it is easy to visualize the point of the maximum stress and displacement. But in real life, parts or assemblies with complex geometric shapes are made up of different materials with many discontinuities subjected to flexible constraints and complex loading varying with respect to time and point of application. This is further complicated by residual stresses and joints like spot and arc welds, etc. Because of this, it is not easy to predict the failure location. Imagine someone shows you a complicated engine block and asks you to predict the failure location for a given set of forces. It is not easy to predict it successfully unless you have years of experience in the field. But with tools like CAD and CAE, if modelled in an appropriate fashion, one can easily get stress contour plots that clearly indicate the locations of high stress or displacement.

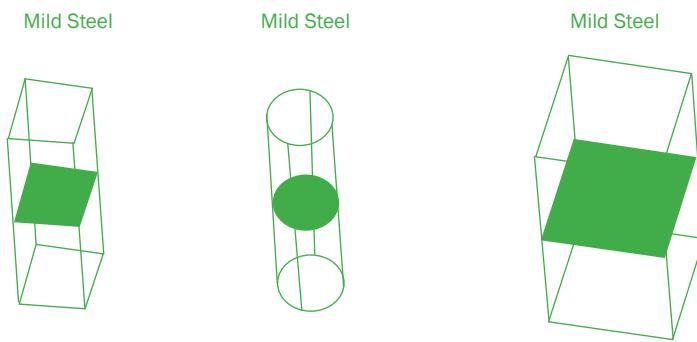
Previously, components used to be designed by highly experienced engineers who had seen a lot of testing and failures of the components in real life. These days, in most organizations, design engineers are very young, using tools like CAD / CAM / CAE and are confident about their designs.

What Is Stiffness And Why Do We Need It In FEA?

Stiffness 'K' is defined as Force/length (units N/mm). Physical interpretation – Stiffness is equal to the force required to produce a unit displacement. The stiffness depends on the geometry as well as the material properties.



Consider 3 rods of exactly the same geometrical dimensions – Cast Iron, Mild Steel, and Aluminium. If we measure the force required to produce a 1 mm displacement then the Cast Iron would require the maximum force, followed by Steel and Aluminium respectively, indicating $K_{CI} > K_{MS} > K_{AI}$



Now consider 3 different cross sectional rods of the same material. Again, the force required to produce a unit deformation will be different. Therefore, stiffness depends on the geometry as well as the material.

Importance of the stiffness matrix - For structural analysis, stiffness is a very important property. The equation for linear static analysis is $[F] = [K] [D]$. The force is usually known, the displacement is unknown, and the stiffness is a characteristic property of the element. This means if we formulate the stiffness matrix for a given shape, like line, quadrilateral, or tetrahedron, then the analysis of any geometry could be performed by meshing it and then solving the equation $F = K D$.

Methods for formulating the stiffness matrix –

- 1) Direct Method
- 2) Variational Method
- 3) Weighted Residual Method

The direct method is easy to understand but difficult to formulate using computer programming. While the Variational and Weighted Residual Methods are difficult to understand, but easy from a programming point of view. That's the reason why all software codes either use the Variational or Weighted Residual Method formulation.

Rod Element Stiffness Matrix Derivation By The Direct Method

Methodology for derivation of stiffness matrix by the direct method:

Assume there are n dof's for a given element (for example, a quad4 element's total dofs = $4 * 6 = 24$).

Step 1) Assume the 1st dof $\neq 0$, and all the other dof = 0. This will lead to equation 1.

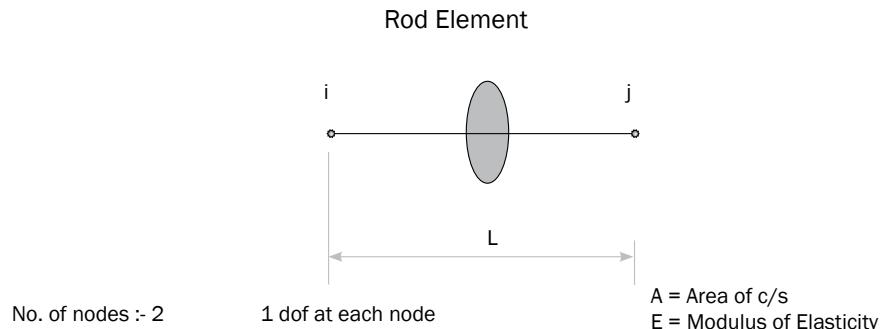
Step 2) Assume the 2nd dof $\neq 0$, and all the other dof = 0. This will lead to equation 2.

:
:

Step n) Assume the nth dof $\neq 0$, and all the other dof = 0. This will lead to equation n.

Step n+1) Sum all the equations, $1 + 2 + 3 + 4 \dots + n$.

Step n+1 will give us the most generalized formulation of the stiffness matrix.



Stiffness matrix formulation (Direct Method)



Case 1:

$$u_i > 0, \quad u_j = 0$$

$$\sum F_x = 0 \quad F_i + F_j = 0 \quad F_i = -F_j$$

$$\sigma_x = F/A \quad \varepsilon = u/L$$

$$\sigma_x = \varepsilon E \quad F/A = Eu/L$$

STATICS

$$\sum F_x = 0 \quad \sum M_x = 0$$

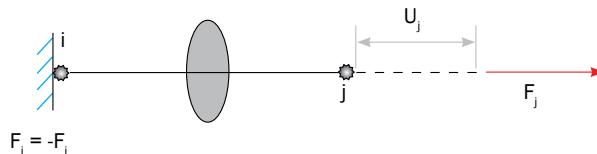
$$\sum F_y = 0 \quad \sum M_y = 0$$

$$\sum F_z = 0 \quad \sum M_z = 0$$

$$F_i = (AE/L)*u_i \quad F_j = -F_i = - (AE/L)*u_i \quad \dots \quad (A)$$

Case 2: $u_i = 0 \quad u_j > 0$

$$F_j = (AE/L)*u_j \quad F_i = -F_j = - (AE/L)*u_j \quad \dots \quad (B)$$



Case 3: General case - $u_i, u_j > 0$

From (A) and (B)

$$F_i = (AE/L)*u_i - (AE/L)*u_j$$

$$F_j = -(AE/L)*u_i + (AE/L)*u_j$$

$$\begin{array}{c}
 \left| \begin{array}{c} F_i \\ F_j \end{array} \right|_{(2 \times 1)} = (AE/L) \left| \begin{array}{cc} 1 & -1 \\ -1 & 1 \end{array} \right|_{(2 \times 2)} \left| \begin{array}{c} u_i \\ u_j \end{array} \right|_{(2 \times 1)}
 \end{array}$$

Force Matrix Stiffness Matrix Displacement Matrix

Properties of the stiffness matrix

- The order of the stiffness matrix corresponds to the total dofs.
- A singular stiffness matrix means the structure is unconstrained and has rigid body motion.
- Each column of the stiffness matrix is an equilibrium set of nodal forces required to produce the unit respective dof.
- A symmetric stiffness matrix shows the force is directly proportional to displacement.
- Diagonal terms of the matrix are always positive meaning a force directed in say the left direction cannot produce a displacement in the right direction. Diagonal terms will be zero or negative only if the structure is unstable.

Rod elements support only tension or compression and no shear force or bending. In the above equation, the order of the stiffness matrix is 2x2, where the number of unknowns is 2.

Number of unknowns = no. of dofs - no. of dofs constraint by a Single Point Constraints (at fixed nodes, dofs are specified by the user as 0)

Usually in comparison to the total dofs for the model, the constraints are negligible and therefore the total no. of unknowns is approximately the total dofs.

The order of the stiffness matrix = total dof x total dof

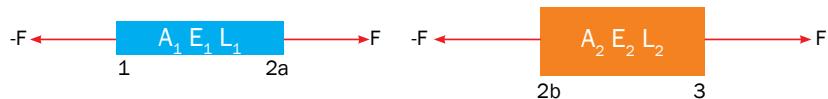
Summary - Stiffness Matrix, Assembly Of 2 Rod Elements



Because of a force at point 3, what would the force be at Points 1 and 2? Before reading the answer, please shut your eyes and try to visualize the forces at the various points.

For equilibrium $\sum F_x = 0$. The reaction force at point 1 is $-F$ and at point 2 = 0.

The free body diagram is:



Total force at point 2 = $-F + F = 0$

In any finite element model, the summation of forces and moments is zero at the internal nodes (except the nodes which are restrained and at which the external force and moment is applied). The overall summation of the forces and the moments for the complete model is 0. (The external forces and moments = the reaction forces and moments). This is one of the important checks for ensuring correct results.

Assembly of elements

$$K_1 = \frac{A_1 E_1}{L_1} \quad \left| \begin{array}{c} 1 & A_1 E_1 L_1 & 2a \\ & & 2b \end{array} \right| \quad K_2 = \frac{A_2 E_2}{L_2} \quad \left| \begin{array}{c} 2b & A_2 E_2 L_2 & 3 \\ & & 3 \end{array} \right|$$

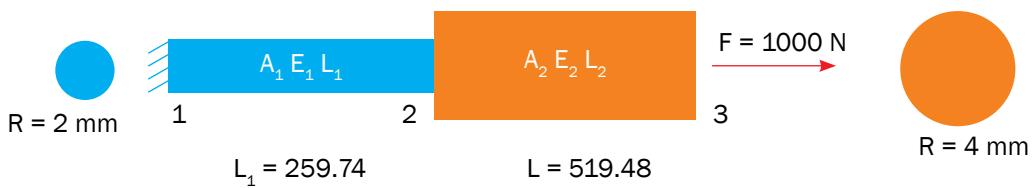
$$\begin{vmatrix} F_1 \\ F_{2a} \\ 0 \end{vmatrix} = \begin{vmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 & 0 \\ 0 & 0 & 0 \end{vmatrix} \begin{vmatrix} u_1 \\ u_{2a} \\ 0 \end{vmatrix} \quad \begin{vmatrix} 0 \\ F_{2b} \\ F_3 \end{vmatrix} = \begin{vmatrix} 0 & 0 & 0 \\ 0 & k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{vmatrix} \begin{vmatrix} 0 \\ u_{2b} \\ u_3 \end{vmatrix}$$

$$\begin{vmatrix} F_1 \\ F_{2a} + F_{2b} \\ F_3 \end{vmatrix} = \begin{vmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1+k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{vmatrix} \begin{vmatrix} u_1 \\ u_{2a} + u_{2b} \\ u_3 \end{vmatrix}$$

$$\begin{vmatrix} F_1 \\ 0 \\ F_3 \end{vmatrix} = \begin{vmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1+k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{vmatrix} \begin{vmatrix} u_1 \\ u_2 \\ u_3 \end{vmatrix}$$

$$\begin{vmatrix} F_1 \\ F_2 \\ F_3 \end{vmatrix}_{3x1} = \begin{vmatrix} k_{11} & k_{12} & k_{13} \\ k_{21} & k_{22} & k_{23} \\ k_{31} & k_{32} & k_{33} \end{vmatrix}_{3x3} \begin{vmatrix} u_1 \\ u_2 \\ u_3 \end{vmatrix}_{3x1}$$

Example: Assembly of 2 rod elements



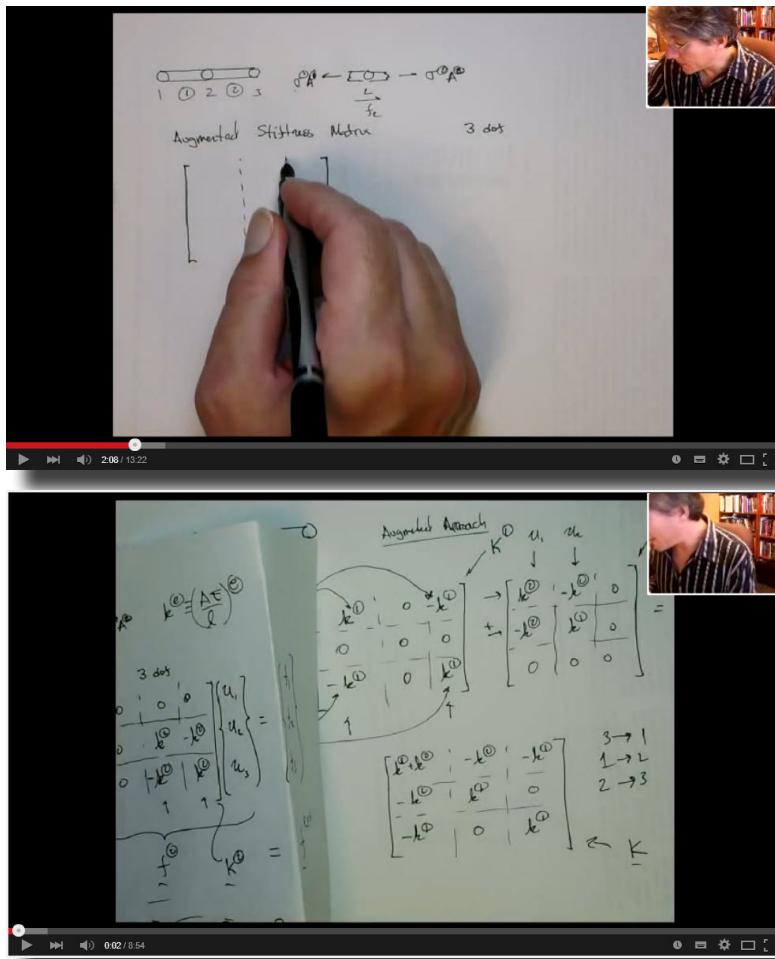
$$K_1 = A_1 E_1 / L_1 = 10^4 \text{ N/mm}$$

$$K_2 = A_2 E_2 / L_2 = 2 * 10^4 \text{ N/mm}$$

$$\begin{vmatrix} F_1 \\ 0 \\ 1000 \end{vmatrix} = 10^4 \begin{vmatrix} 1 & -1 & 0 \\ -1 & 3 & -2 \\ 0 & -2 & 2 \end{vmatrix} \begin{vmatrix} 0 \\ u_2 \\ u_3 \end{vmatrix}$$

$$\begin{vmatrix} 0 \\ 1000 \end{vmatrix} = 10^4 \begin{vmatrix} 3 & -2 & u_2 \\ -2 & 2 & u_3 \end{vmatrix} \begin{array}{l} u_2 = 0.1 \text{ mm} \\ u_3 = 0.15 \text{ mm} \end{array}$$

Some more discussion on assembling the global stiffness matrix from the local stiffness matrices for 1D bar elements.



We need to understand that in the before depicted “Force - Stiffness - Displacement” relationship the global and local coordinate system are the same (element is oriented in x-direction). Typically, they are different. Hence, local and global quantities are “linked” through a transformation matrix as shown below.

For the more general case – which we will have a look at next – local „quantities“ are marked as

$$\begin{bmatrix} F_{\hat{x}1} \\ F_{\hat{x}2} \end{bmatrix} = \frac{E \cdot A}{l} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{bmatrix} \hat{u}_1 \\ \hat{u}_2 \end{bmatrix}$$

$$\hat{K}_e = \frac{E \cdot A}{l} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$\hat{F}_e = \hat{K}_e \cdot \hat{w}_e$$

with

\hat{w}_e = displacement vector, local

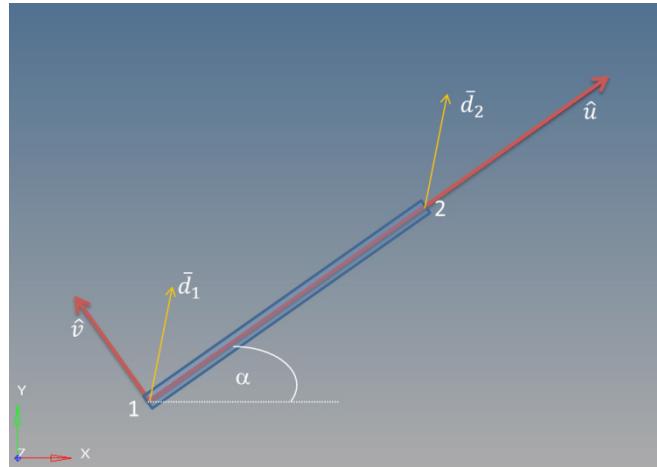
\hat{F}_e = force vector, local

\hat{u}_1, \hat{u}_2 = x – displacements at node 1 and 2, local coordinate system

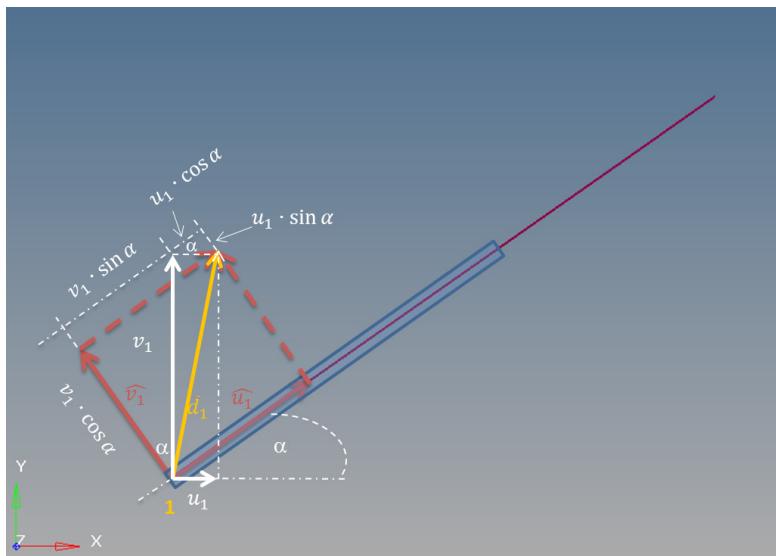
In order to take displacements in y-direction into account

\hat{v}_1, \hat{v}_2 = y – displacements at node 1 and 2, local coordinate system

we are expressing the displacements d in node 1 with respect to the local and global coordinate system.



The displacements in the global coordinate system are written as u_1 and v_1 .



From the image above we can directly derive:

$$\widehat{u}_1 = u_1 \cos \alpha + v_1 \sin \alpha$$

$$\widehat{v}_1 = -u_1 \sin \alpha + v_1 \cos \alpha$$

Written in Matrix form

$$\begin{bmatrix} \widehat{u}_1 \\ \widehat{v}_1 \end{bmatrix} = \begin{bmatrix} \cos \alpha & \sin \alpha \\ -\sin \alpha & \cos \alpha \end{bmatrix} \cdot \begin{bmatrix} u_1 \\ v_1 \end{bmatrix}$$

Including node 2 of the element, the above equation may be written as

$$\begin{bmatrix} \widehat{u}_1 \\ \widehat{v}_1 \\ \widehat{u}_2 \\ \widehat{v}_2 \end{bmatrix} = \begin{bmatrix} \cos \alpha & \sin \alpha & 0 & 0 \\ -\sin \alpha & \cos \alpha & 0 & 0 \\ 0 & 0 & \cos \alpha & \sin \alpha \\ 0 & 0 & -\sin \alpha & \cos \alpha \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{bmatrix}$$

alternatively

$$\widehat{w}_e = T \cdot w_e$$

With T being the transformation matrix: Multiplying global quantities \mathbf{w}_e with T transforms them into local quantities $\widehat{\mathbf{w}}_e$.

Nodal forces may be written in the same way:

$$\begin{bmatrix} F_{\bar{x}1} \\ F_{\bar{y}1} \\ F_{\bar{x}2} \\ F_{\bar{y}2} \end{bmatrix} = \begin{bmatrix} \cos \alpha & \sin \alpha & 0 & 0 \\ -\sin \alpha & \cos \alpha & 0 & 0 \\ 0 & 0 & \cos \alpha & \sin \alpha \\ 0 & 0 & -\sin \alpha & \cos \alpha \end{bmatrix} \begin{bmatrix} F_{x1} \\ F_{y1} \\ F_{x2} \\ F_{y2} \end{bmatrix}$$

alternatively

$$\widehat{\mathbf{F}}_e = T \cdot \mathbf{F}_e$$

Quite obviously, if global quantities are requested, we need to rearrange the above equation.

$$T^{-1} \cdot \widehat{\mathbf{F}}_e = T^{-1} \cdot T \cdot \mathbf{F}_e = E \cdot \mathbf{F}_e = \mathbf{F}_e$$

Here it is

$$T^{-1} = T^T$$

thus

$$\mathbf{F}_e = T^T \cdot \widehat{\mathbf{F}}_e$$

As we already know, multiplying local quantities with T^T transforms into global quantities.

More generally, we include the nodal forces $F_{\bar{y}1}$ (node 1) and $F_{\bar{y}2}$ (node 2) in our equation from above (here $F_{\bar{y}1}$ and $F_{\bar{y}2}$ are 0).

$$\begin{bmatrix} F_{\bar{x}1} \\ F_{\bar{y}1} \\ F_{\bar{x}2} \\ F_{\bar{y}2} \end{bmatrix} = \frac{E \cdot A}{l} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} \widehat{u}_1 \\ \widehat{v}_1 \\ \widehat{u}_2 \\ \widehat{v}_2 \end{bmatrix}$$

$$\widehat{\mathbf{F}}_e = \widehat{\mathbf{K}}_e \cdot \widehat{\mathbf{w}}_e$$

Taking into account

$$\widehat{\mathbf{w}}_e = T \cdot \mathbf{w}_e$$
 from above follows

$$\widehat{\mathbf{F}}_e = \widehat{\mathbf{K}}_e \cdot T \cdot \mathbf{w}_e$$

And again making use of

$$T^{-1} = T^T$$

and

$$\mathbf{F}_e = T^T \cdot \widehat{\mathbf{F}}_e$$

results in

$$\mathbf{F}_e = T^T \cdot \widehat{\mathbf{K}}_e \cdot T \cdot \mathbf{w}_e$$

or simply

$$\mathbf{F}_e = \mathbf{K}_e \cdot \mathbf{w}_e$$

with

$$\mathbf{K}_e = T^T \cdot \widehat{\mathbf{K}}_e \cdot T$$

Let's have a look at the K_e matrix

$$K_e = \frac{E \cdot A}{l} \cdot \begin{bmatrix} \cos^2 \alpha & \cos \alpha \sin \alpha & -\cos^2 \alpha & -\cos \alpha \sin \alpha \\ \cos \alpha \sin \alpha & \sin^2 \alpha & -\cos \alpha \sin \alpha & -\sin^2 \alpha \\ -\cos^2 \alpha & -\cos \alpha \sin \alpha & \cos^2 \alpha & \cos \alpha \sin \alpha \\ -\cos \alpha \sin \alpha & -\sin^2 \alpha & \cos \alpha \sin \alpha & \sin^2 \alpha \end{bmatrix}$$

The [] matrix is commonly „abbreviated“ as

$$\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix}$$

Example

Let's apply these basics to the following „problem“ which was originally formulated by Professor B. Wender (University Ulm / Germany).

Objective: Determine the nodal displacements and the force in element I

Given:

$$a = 810 \text{ mm}$$

$$b = 480 \text{ mm}$$

$$c = 680 \text{ mm}$$

$$\alpha = 37^\circ$$

$$\beta = 18^\circ$$

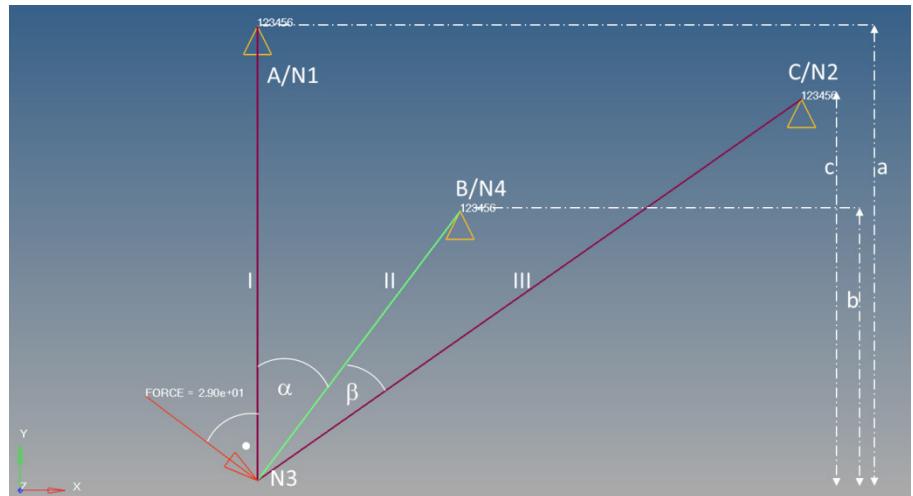
$$F = 29 \text{ kN}$$

$$F_x = F \cdot \cos \alpha = 23,160 \text{ kN}$$

$$F_y = -F \cdot \sin \alpha = -17,453 \text{ kN}$$

$$A^I = A^{II} = A^{III} = 280 \text{ mm}^2$$

$$E^I = E^{III} = 210 \text{ kN/mm}^2 \quad E^{II} = 70 \text{ kN/mm}^2$$



Let us first determine the values of $D^e = \frac{E^e \cdot A^e}{l^e}$ regarding every element, respectively.

$$D^I = \frac{E^I \cdot A^I}{l^I} = \frac{210 \cdot 280}{810} = 72,593$$

$$D^{II} = \frac{E^{II} \cdot A^{II}}{l^{II}} = \frac{70 \cdot 280}{601,03} = 32,611$$

$$D^{III} = \frac{E^{III} \cdot A^{III}}{l^{III}} = \frac{210 \cdot 280}{1185,54} = 49,598$$

Units in kN and mm.

Element I

$$\alpha = -90^\circ$$

$$c^2 = 0; s^2 = 1; cs = 0$$

$$D^I = 72,593$$

$$F^I = K^I \cdot w^I = D^I \cdot \begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \cdot \begin{bmatrix} u_1 \\ v_1 \\ u_3 \\ v_3 \end{bmatrix} = \begin{bmatrix} F^I_{x1} \\ F^I_{y1} \\ F^I_{x3} \\ F^I_{y3} \end{bmatrix}$$

$$\begin{bmatrix} F^I_{x1} \\ F^I_{y1} \\ F^I_{x3} \\ F^I_{y3} \end{bmatrix} = \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 72,593 & 0 & -72,593 \\ 0 & 0 & 0 & 0 \\ 0 & -72,593 & 0 & 72,593 \end{bmatrix} \cdot \begin{bmatrix} u_1 \\ v_1 \\ u_3 \\ v_3 \end{bmatrix}$$

Element II

$$\alpha = 53_0$$

$$c^2 = 0,3622; s^2 = 0,6378; cs = 0,4806$$

$$D^{II} = 32,611$$

$$F^{II} = K^{II} \cdot w^{II} = D^{II} \cdot \begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \cdot \begin{bmatrix} u_3 \\ v_3 \\ u_4 \\ v_4 \end{bmatrix} = \begin{bmatrix} F^{II}_{x3} \\ F^{II}_{y3} \\ F^{II}_{x4} \\ F^{II}_{y4} \end{bmatrix}$$

$$\begin{bmatrix} F^{II}_{x3} \\ F^{II}_{y3} \\ F^{II}_{x4} \\ F^{II}_{y4} \end{bmatrix} = \begin{bmatrix} 11,812 & 15,673 & -11,812 & -15,673 \\ 15,673 & 20,799 & -15,673 & -20,799 \\ -11,812 & -15,673 & 11,812 & 15,673 \\ -15,673 & -20,799 & 15,673 & 20,799 \end{bmatrix} \cdot \begin{bmatrix} u_3 \\ v_3 \\ u_4 \\ v_4 \end{bmatrix}$$

Element III

$$\alpha = -145_0$$

$$c^2 = 0,6710; s^2 = 0,3280; cs = 0,4698$$

$$D^{III} = 49,598$$

$$F^{III} = K^{III} \cdot w^{III} = D^{III} \cdot \begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \cdot \begin{bmatrix} u_2 \\ v_2 \\ u_3 \\ v_3 \end{bmatrix} = \begin{bmatrix} F^{III}_{x2} \\ F^{III}_{y2} \\ F^{III}_{x3} \\ F^{III}_{y3} \end{bmatrix}$$

$$\begin{bmatrix} F^{III}_{x2} \\ F^{III}_{y2} \\ F^{III}_{x3} \\ F^{III}_{y3} \end{bmatrix} = \begin{bmatrix} 33,281 & 23,303 & -33,281 & -23,303 \\ 23,303 & 16,317 & -23,303 & -16,317 \\ -33,281 & -23,303 & 33,281 & 23,303 \\ -23,303 & -16,317 & 23,303 & 16,317 \end{bmatrix} \cdot \begin{bmatrix} u_2 \\ v_2 \\ u_3 \\ v_3 \end{bmatrix}$$

Force equilibrium

$$F^I_{x1} = F_{Ax}$$

$$F^I_{y1} = F_{Ay}$$

$$F^{II}_{x4} = F_{Bx}$$

$$F^{II}_{y4} = F_{By}$$

$$F^{III}_{x4} = F_{Cx}$$

$$F^{III}_{y4} = F_{Cy}$$

$$F_x = F^I_{x3} + F^{II}_{x3} + F^{III}_{x3}$$

$$F_y = F^I_{y3} + F^{II}_{y3} + F^{III}_{y3}$$

With above the system of equations may be written as:

$$\underbrace{\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 72,593 & 0 & 0 & 0 & -72,593 & 0 & 0 \\ 0 & 0 & 33,281 & 23,303 & -33,281 & -23,303 & 0 & 0 \\ 0 & 0 & 23,303 & 16,317 & -23,303 & -16,317 & 0 & 0 \\ 0 & 0 & -33,281 & -23,303 & 45,092 & 38,976 & 11,812 & -15,673 \\ 0 & -72,593 & -23,303 & -16,317 & 38,976 & 109,701 & -15,673 & 20,799 \\ 0 & 0 & 0 & 0 & -11,811 & -15,674 & 11,811 & 15,673 \\ 0 & 0 & 0 & 0 & -15,674 & -20,800 & 15,674 & 20,800 \end{bmatrix}}_{\text{Node 1}} \cdot \underbrace{\begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{bmatrix}}_{\text{Node 2}} = \underbrace{\begin{bmatrix} F_{Ax} \\ F_{Ay} \\ F_{Cx} \\ F_{Cy} \\ F_x \\ F_y \\ F_{Dx} \\ F_{Dy} \end{bmatrix}}_{\text{Node 3}} \cdot \underbrace{\begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{bmatrix}}_{\text{Node 4}}$$

Taking the constraints:

$$u_1 = v_1 = u_2 = v_2 = u_4 = v_4 = 0$$

into account leads to the 2 equations

$$45,092 \cdot u_3 + 38,976 \cdot v_3 = 23,160$$

and

$$38,976 \cdot u_3 + 109,701 \cdot v_3 = -17,453$$

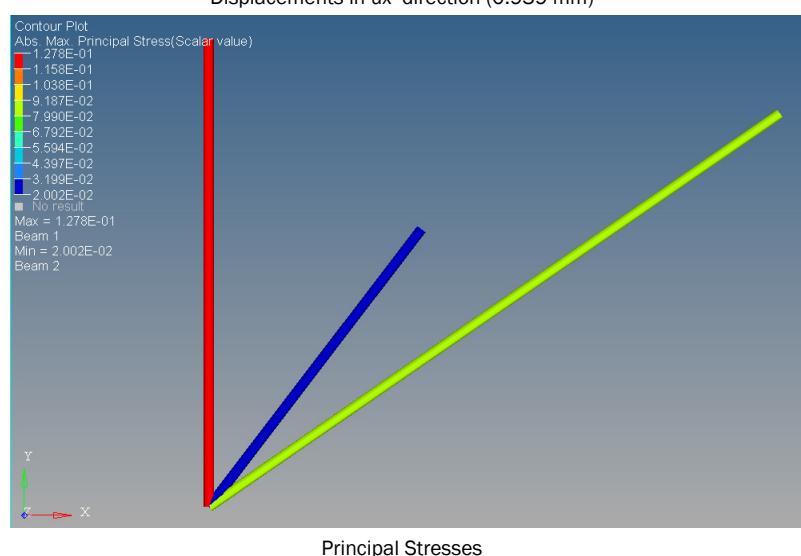
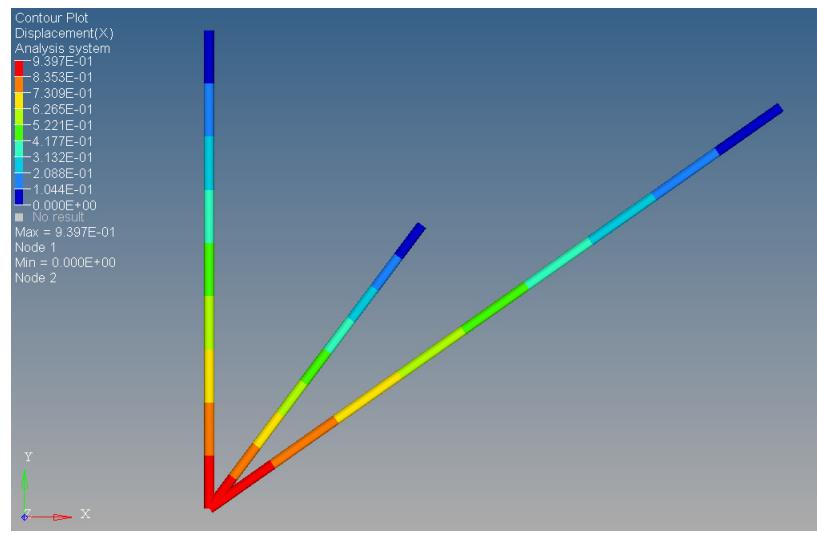
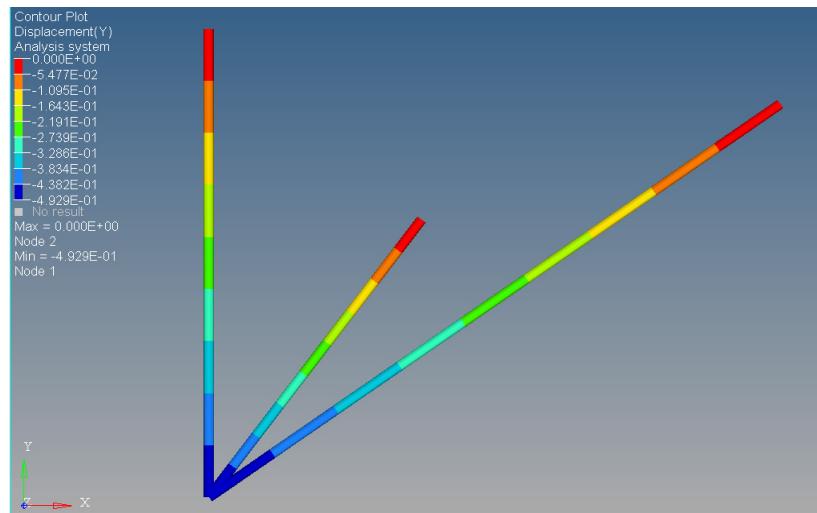
$$v_3 = -0,493 \text{ mm}$$

$$u_3 = 0,940 \text{ mm}$$

Foce in element I (with values from above)

$$F^I = A^I \cdot E^I \cdot \epsilon^I = 35,784 \text{ kN}$$

The corresponding simulation results based on OptiStruct are shown below. Note: The model set-up and analysis are discussed in Chapter „Linear Static Analysis“.



Force in element I

$$F^I = \sigma \cdot A = 0,1278 \cdot 280 = 35,784 \text{ kN}$$

In case you are not experienced with truss structures you may like the lesson (video below) on solving a truss (not related to FEA).

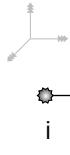
Topics covered include determining whether the truss is statically determinate, finding external reaction forces, and finding internal forces (<https://www.youtube.com/watch?v=qzmeFq8rckw>)



Beam Element

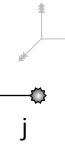
The beam element is the most general type of element (6 dofs per node, 3 translational and 3 rotational) and is used frequently for various applications.

123456



2 Nodes

123456



6 dof per node (3T + 3R)

$$[F]_{12 \times 1} = [K]_{12 \times 12} [\delta]_{12 \times 1}$$

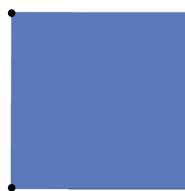
F_{ix}		$K_{11} \quad K_{12} \quad K_{13} \quad K_{14} \quad K_{15} \quad K_{16} \quad K_{17} \quad K_{18} \quad K_{19} \quad K_{110} \quad K_{111} \quad K_{112}$		u_{ix}
F_{iy}		$K_{22} \quad K_{23} \quad K_{24} \quad K_{25} \quad K_{26} \quad K_{27} \quad K_{28} \quad K_{29} \quad K_{210} \quad K_{211} \quad K_{212}$		v_{iy}
F_{iz}		$K_{33} \quad K_{34} \quad K_{35} \quad K_{36} \quad K_{37} \quad K_{38} \quad K_{39} \quad K_{310} \quad K_{311} \quad K_{312}$		w_{iz}
M_{ix}		$K_{44} \quad K_{45} \quad K_{46} \quad K_{47} \quad K_{48} \quad K_{49} \quad K_{410} \quad K_{411} \quad K_{412}$		θ_{ix}
M_{iy}		$K_{55} \quad K_{56} \quad K_{57} \quad K_{58} \quad K_{59} \quad K_{510} \quad K_{511} \quad K_{512}$		θ_{iy}
M_{iz}		$K_{66} \quad K_{67} \quad K_{68} \quad K_{69} \quad K_{610} \quad K_{611} \quad K_{612}$		θ_{iz}
F_{jx}	=	$K_{77} \quad K_{78} \quad K_{79} \quad K_{710} \quad K_{711} \quad K_{712}$		u_{jx}
F_{jy}		$K_{88} \quad K_{89} \quad K_{810} \quad K_{811} \quad K_{812}$		v_{jy}
F_{jz}		$K_{99} \quad K_{910} \quad K_{911} \quad K_{912}$		w_{jz}
M_{jx}		$K_{1010} \quad K_{1011} \quad K_{1012}$		θ_{jx}
M_{jy}		$K_{1111} \quad K_{1112}$		θ_{jy}
M_{jz}		K_{1212}		θ_{jz}
	12x1		12x12	12x1

* Beam element can resist force and moment about x, y, and z axis.

What About 2D Elements?

Recall: The order of the stiffness matrix = total dof x total dof

Quad4 element:

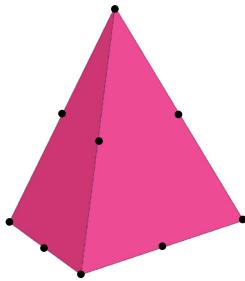


dof/node = 6
total dofs = 6X4 = 24,
order of the stiffness matrix = 24 x 24

$$|F|_{24x1} = |K|_{24x24} |\delta|_{24x1}$$

This means to solve a single quad4 element problem, the software will have to solve 24 equations internally.

Tetra10 element:



dof /node = 3
total dofs = 3*10 = 30
order of stiffness matrix = 30 x30

$$|F|_{30x1} = |K|_{30x30} |\delta|_{30x1}$$

For A Given FE Model, How Many Equations Does A Software Solve?

Assume there are 20,000 nodes for a mesh model consisting of only thin shell elements (6 dof/node).

Total dofs = 20000*6 = 120,000.

Stiffness matrix order = 120,000x120,000

Number of equations the FEA software will solve internally = 120,000

Table On The Equivalence Of Variational FEM And Weighted Residual FEM In Solid Mechanics

The problem: A one dimensional rod element is subjected to a concentrated tensile force at one end (free end) and the other end is fixed as discussed above.

The governing differential equation and boundary conditions:

$$AE \left(\frac{d^2u}{dx^2} \right) = 0 \quad u|_{x=0} = 0$$

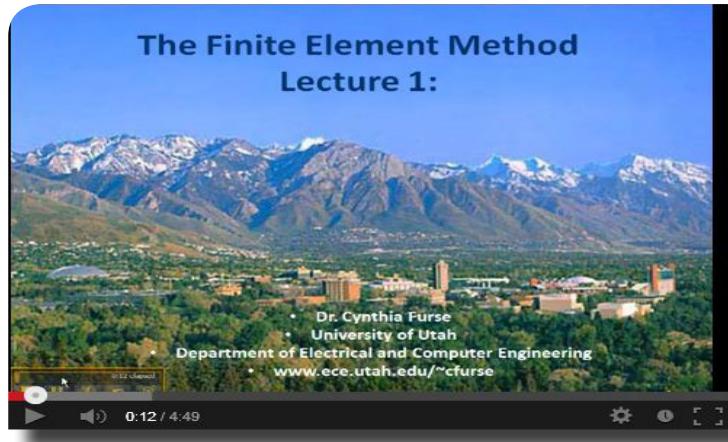
$$AE \left(\frac{du}{dx} \right)|_{x=L} = P$$

Rayleigh-Ritz FEM	Galerkin FEM
1. Starting point $\Pi = \int_{x_i}^{x_j} [AE/2 (dy/dx)^2 dx] - P u_j$	Strong form $AE(d^2u/dx^2) = 0$ Weak form $\int_{x_i}^{x_j} w_i [AE (d^2u/dx^2)] dx = 0$ w_i = Weighting function Integrating by parts $AE \int_{x_i}^{x_j} dw_i/dx du/dx dx = [w_i AE du/dx]_0^L$
2. FE discretization step $u = \sum u_i N_i = u_i N_i + u_j N_j$ $N_i = (x_j - x_i)/L, N_j = (x_i - x_j)/L$	Where $L = (x_j - x_i)$ $du/dx = (u_j - u_i)/L$
3. Substituting the derivative in the functional $\Pi = AE/2 \int_{x_i}^{x_j} (u_j - u_i/L)^2 dx - P u_j$	3. Galerkin step $AE \int_{x_i}^{x_j} dN_i/dx dN_j/dx dx \begin{bmatrix} U_i \\ U_j \end{bmatrix} = \begin{bmatrix} N_i AE du/dx \end{bmatrix}_{x_i}^{x_j}$
4. Minimizing the functional $\Pi = AE/2L[u_i^2 - 2u_i u_j + u_j^2] - P u_j$ $\frac{\partial \Pi}{\partial u_i} = AE/2L[2u_i - 2u_j] = 0$ $\frac{\partial \Pi}{\partial u_j} = AE/2L[-2u_i + 2u_j] - P = 0$	4. Evaluate the element stiffness matrix $K_e = AE \int_{x_i}^{x_j} dN_i/dx dN_j/dx dx$ $= AE \begin{bmatrix} \int_{x_i}^{x_j} dN_i/dx dN_j/dx dx & \int_{x_i}^{x_j} dN_i/dx dN_j/dx dx \\ \int_{x_i}^{x_j} dN_j/dx dN_i/dx dx & \int_{x_i}^{x_j} dN_j/dx dN_i/dx dx \end{bmatrix}$ $= AE / L \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$ (as $dN_i/dx = -1/L$, $dN_j/dx = 1/L$)
5. Write down the final algebraic form of equations $= AE/2L \begin{bmatrix} 2 & -2 \\ -2 & 2 \end{bmatrix} \begin{bmatrix} U_i \\ U_j \end{bmatrix} = \begin{bmatrix} 0 \\ P \end{bmatrix}$ $= AE/L \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{bmatrix} U_i \\ U_j \end{bmatrix} = \begin{bmatrix} -AE du/dx _{x=x_i} \\ AE du/dx _{x=x_j} \end{bmatrix} \begin{bmatrix} 0 \\ P \end{bmatrix}$	5. Write down the final algebraic form $= AE/L \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{bmatrix} U_i \\ U_j \end{bmatrix} = \begin{bmatrix} -AE du/dx _{x=x_i} \\ AE du/dx _{x=x_j} \end{bmatrix} \begin{bmatrix} 0 \\ P \end{bmatrix}$
Observation: Both of the formulations lead to the same final algebraic form of the stiffness matrix	

3.2 Opensource FEM Courses

There are literally thousands of recorded FEM lectures available on the web (especially Youtube).

For instance, the lecture by **Dr. Cynthia Furse**, University of Utah (related to electromagnetic field simulation http://youtu.be/9uPxFZ_T3ok)



or the lecture series by **Prof. K.J. Bathe** (<http://youtu.be/20WSeL4tz2k>)

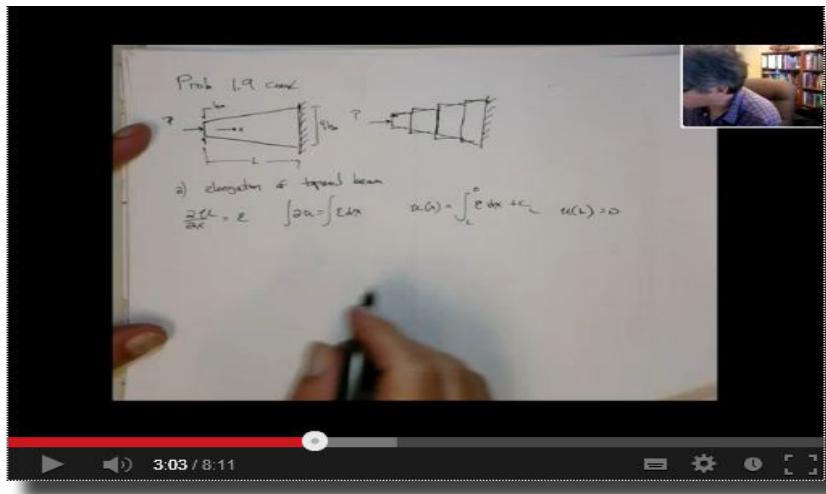


This video series is a comprehensive course of study that presents effective finite element procedures for the linear analysis of solids and structures.

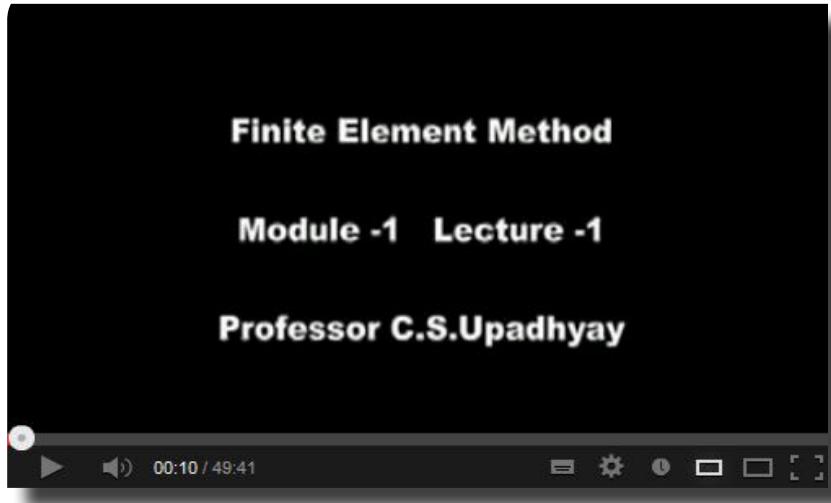
Professor K. J. Bathe, teaches the basic principles used for effective finite element analysis, describes the general assumptions, and discusses the implementation of finite element procedures. Upon completion of this video course, a second course covering Nonlinear Analysis is available.



Professor Chessa, University of Texas, El Paso (http://youtu.be/CBypsx_u3M8)



Prof. C.S.Uppadhyay Department of Aero Space IIT Kanpur (<http://youtu.be/NYiZQszx9cQ>)



A brief summary on FEM theory provided to students participating in the BAJA 2013/2014 Challenge (<http://altair-2.wistia.com/medias/lfkklcibb0>)

The page has a header 'What is FEM?' and the Altair logo. It features two diagrams: one showing a circle with points and another showing a circle divided into triangles. Below the diagrams, text states:

No. of Points = ∞
dof per Point = 6
Total equations = ∞

No. of Nodes = 8
dof per Node = 6
Total equations = 48

FEM (Finite Element Method)

- o A numerical method.
- o Mathematical representation of an actual problem.
- o Approximate method.

The Finite Element Method only makes calculations at a limited (Finite) number of points and then interpolates the results for the entire domain (surface or volume).

Finite – Any continuous object has infinite degrees of freedom and it is not possible to solve the problem in this format. The Finite Element Method reduces the degrees of freedom from infinite to finite with the help of discretization or meshing (nodes and elements).

Element – All of the calculations are made at a limited number of points known as nodes. The entity joining nodes and forming a specific shape such as quadrilateral or triangular is known as an Element. To get the value of a variable (say displacement) anywhere in between the calculation points,

You may also visit the website of University of Colorado at Boulder

Introduction to Finite Element Methods (ASEN 5007)
Fall 2014
Department of Aerospace Engineering Sciences
University of Colorado at Boulder

This is the public web site for the graduate core course ASEN 5007: **Introduction To Finite Element Methods** (IFEM). This **master level** course is part of the **Aerospace Systems Focus Area** of the graduate curriculum in the Department of Aerospace Engineering Sciences of the University of Colorado at Boulder. Offered yearly during the **Fall Semester**, both on-campus and remotely through CAETE. First taught in 1987. This website dates from 1998 and is continuously being revised. Related courses may be accessed at:

AFEM: Advanced Finite Element Methods (ASEN 6347) Master & doctoral level
AVMM: Advanced Variational Methods in Mechanics Master & doctoral level (in preparation)
FSI: Fluid Structure Interaction (ASEN 5509) Doctoral level
MFEMD: Matrix Finite Element Methods in Dynamics Master & senior-elective level (in preparation)
MFEML: Matrix Finite Element Methods in Statics Master & senior-elective level (in preparation)
NFEM: Nonlinear Finite Element Methods (ASEN 6107) Master & doctoral level
IAST: Intro to Aerospace Structures (ASEN 3112) Junior undergraduate level

[General Course Information](#). Syllabus, coursework, schedule, roster ...

Part 0: Preface
[Preface Index](#).

Part I: Finite Element Discretization and the Direct Stiffness Method

[Chapter 1 Index](#). Overview. Not covered
[Chapter 2 Index](#). The Direct Stiffness Method I.
[Chapter 3 Index](#). The Direct Stiffness Method II.
[Chapter 4 Index](#). Analysis of Example Truss by a CAS.
[Chapter 5 Index](#). Constructing MoM Members.
[Chapter 6 Index](#). Finite Element Modeling: Introduction.
[Chapter 7 Index](#). Finite Element Modeling: Mesh, Loads, BCs.
[Chapter 8 Index](#). Multifreedom Constraints I.
[Chapter 9 Index](#). Multifreedom Constraints II.

<http://www.colorado.edu/engineering/CAS/courses.d/IFEM.d/Home.html>

Introduction to Finite Element Method by Dr. R. Krishnakumar, Department of Mechanical Engineering, IIT Madras. This introduction consists of a series of 33 videos. For more details on NPTEL visit <http://nptel.ac.in>

Mechanical - Introduction to Finite Element Method

von nptelhrd • 1/33 Videos

Mod-01 Lec-01 Introduction to Finite Element Method
Mod-01 Lec-02 Introduction to Finite Element Method
Mod-01 Lec-03 Introduction to Finite Element Method
Mod-01 Lec-04 Introduction to Finite Element Method
Mod-01 Lec-05 Introduction to Finite Element Method
Mod-01 Lec-06 Introduction to Finite Element Method
Mod-01 Lec-07 Introduction to Finite Element Method

<https://www.youtube.com/watch?v=KR74TQesUoQ&list=PLbMVogVj5nJRjnZA9oryBmDdUNe7lbnB0>

Again, these are just a few classes we came across by looking for public FEA classes. In case you like to add (or recommend) other classes, please let us know.

Just drop a mail to altairuniversity@altair.com

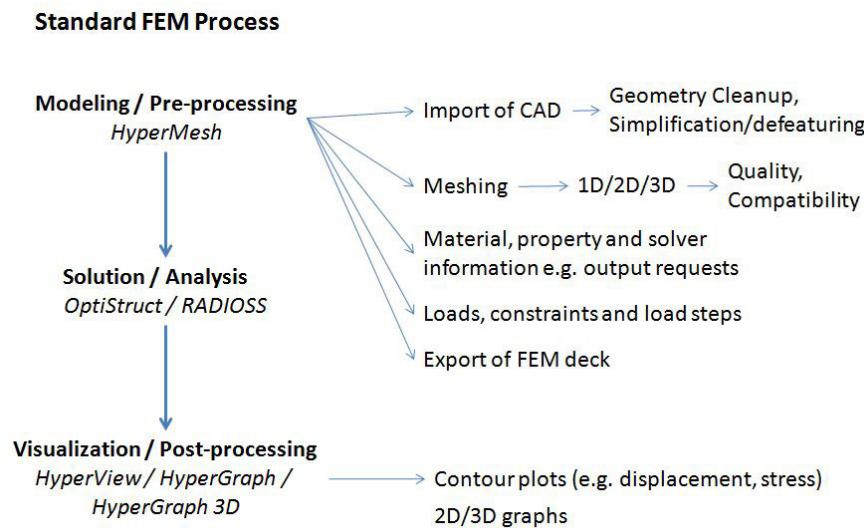
4 What Is Needed To Run A Finite Element Analysis?

This chapter includes material from the book “**Practical Finite Element Analysis**”. It also has been reviewed and has additional material included by Matthias Goelke and Jan Grasmannsdorf.

4.1 Basic Information Needed To Run A Finite Element Analysis

In a high level summary, the “working” steps involved in a finite element analysis may be categorized as:

- Modeling (pre-processing)
- Solution
- Visualization of solution results (post-processing)



This image depicts the three elementary working steps involved in a FEM analysis. Some details about the individual steps are summarized below.

4.2 Modeling / Pre-Processing

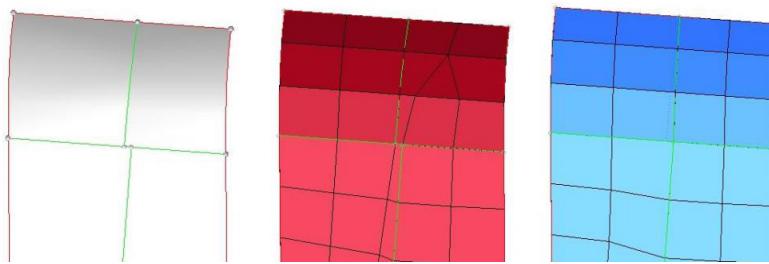
CAD Data

Most commonly, an FEM simulation process starts with the import of the component’s (or part’s) CAD geometry (e.g. CATIA, STEP, UG, IGES, solidThinking, etc.) into the pre-processor i.e. HyperMesh

In many cases, the imported geometry is not ready for meshing. Quite often the geometry needs a cleanup first due to

- “broken” surfaces
- surfaces which are not stitched together
- redundant (multiple) surfaces
- surfaces which are too small to be meshed in a reasonable way later on

Another issue related to geometry is depicted in the following image:



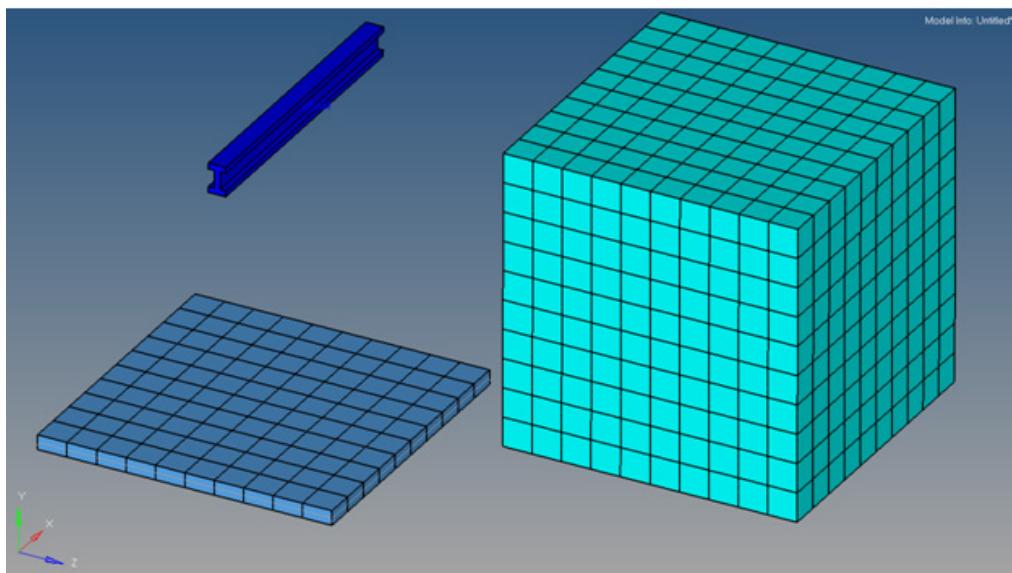
In the image on the left, the imported geometry is shown. Note the lateral offset of the green edges. Here, the surface edges (in green) do not meet at a single point i.e. there is a very small lateral offset of the surface edges. As meshing is carried out with respect to the surfaces, this small offset will be automatically taken into account during meshing, which, unfortunately will result in very poor quality elements. The image in the middle depicts the meshed “initial” geometry. Note how the mesh is locally distorted. The updated (cleaned) and meshed geometry is shown on the right.

Here, the surface edges (in green) do not meet in a single point, i.e. there is a very small lateral offset of the surface edges. As meshing is carried out with respect to the surfaces, this small offset will be automatically taken into account during meshing, which, unfortunately will result in very poor quality elements.

Once these “hurdles” are mastered, one needs to ask whether all the CAD information is really needed. What about little fillets and rounds, tiny holes or even company logos which can often be found in CAD data? Do they really contribute to the overall performance of the component?

Meshing

Once the geometry is in an appropriate state, a mesh is created to approximate the geometry. Either a beam mesh (1D), shell mesh (2D) or a solid mesh (3D) will be created. This meshing step is crucial to the finite element analysis as the quality of the mesh directly reflects on the quality of the results generated. At the same time, the number of elements (number of nodes) affects the computation time. That is the reason why in certain cases a 2D and 1D mesh is preferred over 3D mesh. For example, in sheet metals a 2D approximation of the structure uses much less elements and thus reduces the CPU time (which is the time while you are waiting for your results).



See the picture above for structures that are typically meshed with 1D, 2D and 3D elements. Which element type would you choose for which part?

Despite the fact that meshing is (at least optionally) a highly automated process, mesh quality, its connectivity (i.e. compatibility), and element normals needs to be checked. If necessary, these element “issues” may need to be improved by updating (altering) the underlying geometry or by editing single elements.

Material And Property Information

After meshing is completed, material (e.g. Young's Modulus) and property information (e.g. thickness values) are assigned to the elements.

Loads, Constraints And Solver Information

Various loads and constraints are added to the model to represent the loading conditions that the part(s) are subjected to. Different load cases can be defined to represent different loading conditions on the same model. Solver information is also added to tell the solver what kind of analysis is being run, which results to export, etc.

To determine your relevant loads, your engineering skills are needed. Think of all kinds of load situations that can occur on your structure and decide whether you want to use them in your simulation or not. To determine the load from a static or dynamic event, a Multibody Simulation (MBD) might be helpful.

The FEM model (consisting of nodes, elements, material properties, loads and constraints) is then exported from within the pre-processor HyperMesh. The exported FEM model, typically called solver input deck, is an ASCII file based on the specific syntax of the FEM solver chosen for the analysis (e.g. RADIOSS or OptiStruct). A section out of an OptiStruct solver deck is depicted in the figure below.

```

$$ Optistruct Input Deck Generated by HyperMesh Version : 10.0build60
$$ Generated using HyperMesh-Optistruct Template Version : 10.0-SA1-110
$$
$$ Template: optistruct
$$
$$
$$
$$
$$-----$ Case Control Cards $-----$
$$
$$HMNAME LOADSTEP           1"loadstep"          0
$$
SUBCASE      1
SPC =        1
LOAD =       2
$$-----$ HYPERMESH TAGS $-----$
$$-----$ BEGIN TAGS $-----$
$$-----$ END TAGS $-----$
$ 
BEGIN BULK
$$
$$ Stacking Information for Ply-Based Composite Definition
$$

PARAM,CHECKEL,NO
$$
$$ DESVARG Data
$$
$$
$$ GRID Data
$$
GRID      65      100.0    80.0   -2.17-10
GRID      66      100.0    77.5   -2.2-10
GRID      67      100.0    55.0   -2.2-10

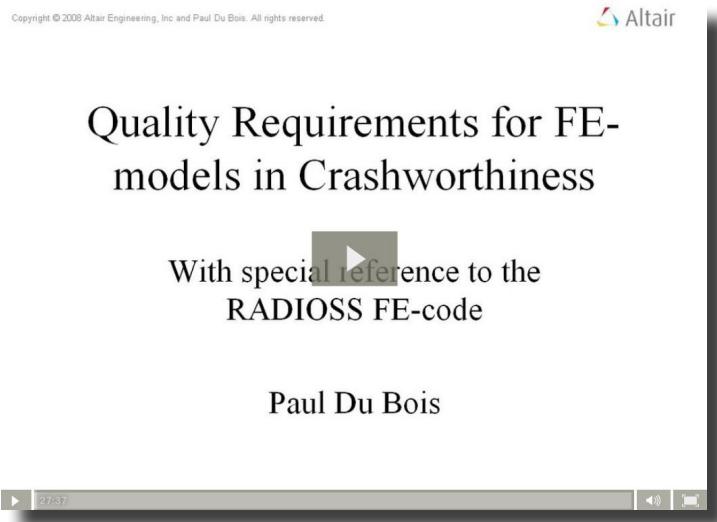
```

As you will see, the bulk of information stored in the analysis file is related to the definition of nodes (or grids). Each single node is defined by its nodal number (ID) and its x-, y- and z coordinates. Each element is then defined by its element number (ID) and its nodes (IDs are referenced). This completes the pre-processing phase.

4.3 Solution

During the solution phase of a simple linear static analysis or an eigenfrequency study, there is not much for you to do. The default settings of the Finite Element program do handle these classes of problems pretty well. Practice will show you that if the solution process is aborted by an “error”, it is due to mistakes you have made during the model building phase. Just to mention a few typical errors:

- Element quality (<http://altair-2.wistia.com/medias/rmretouym>)



- Invalid material properties
- Material property not assigned to the elements
- Insufficiently constrained model (the model shows a rigid body motion due to external loads)

Some of these model issues are discussed in additional videos/webinars available on the **Academic Training Center** (<http://training.altairuniversity.com/>)

4.4 Visualization / Post-Processing

Once the solution has ended successfully, post-processing (in HyperView for contour plots and HyperGraph for 2D/3D plots) of the simulation results is done next. Stresses, strains, and deformations are plotted and examined to see how the part responded to the various loading conditions. Based on the results, modifications may be made to the part and a new analysis may be run to examine how the modifications affected the part.

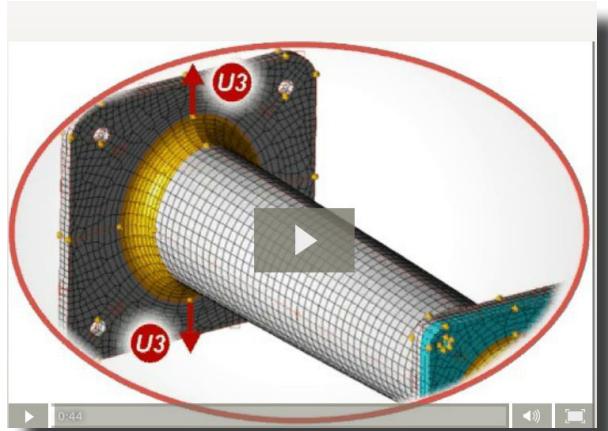
This eventually completes the FEM process.

Remarks

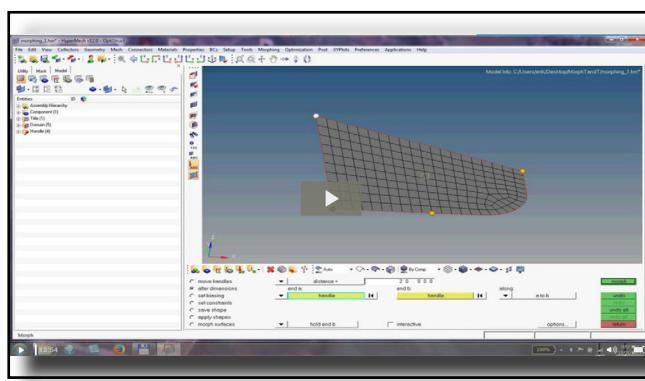
Practice will show, that in many projects, the above depicted process must be re-entered again, because simulation results indicate that the part is not performing as requested. It is quite obvious that going back to CAD (to apply changes) and working through the entire FEM process becomes tedious.

A very efficient (and exciting) technology to speed up this process is called **Morphing**. Employing morphing allows the CAE engineer to modify the geometry of the FEM model, e.g. change radii, thickness of ribs, shape of hard corners, etc. Quite often the morphed FEM model can be exported instantaneously (without any remeshing) allowing the CAE engineer to re-run the analysis of the modified part on the fly.

An example of morphing a given finite element model is depicted below (<http://altair-2.wistia.com/medias/dp9q29f3jn>)



A nice introduction in morphing is given in the video below (<http://altair-2.wistia.com/medias/xycj10c4y2>).



Note:

The individual working steps of the FEM process are not only subjected to many “user” errors e.g. typo while defining material or loads. A lot of attention must also be paid to the chosen modeling assumptions (for instance, simplification of geometry, chosen element type and size, etc.). Even though the FEM solver may detect some of the most striking errors, the likelihood that your results have bypassed “errors” is high.

The following chapters aim at creating awareness about FEM challenges and pitfalls.

5 Strategic Planning

This chapter includes material from the book “**Practical Finite Element Analysis**”. It also has been reviewed and has additional material included by Matthias Goelke.

5.1 Planning The Solution

Planning the various phases of a problem’s solution is the first step in solving an engineering problem. Stay focused on the problem itself rather than on mechanical processes to ensure that the model you create accurately represents the problem in question. Thoughtful planning will save time in the long run and ensure a higher level of success.

Understanding the relationship between your solution strategy and the problem objectives will help you fulfil your responsibility to provide the customer with information concerning assumptions you have made and the limitations of methods used.

The following sections provide an outline of issues to consider prior to modeling, which will help to plan a problem’s solution.

Communicate

Work toward understanding the customer’s needs by maintaining a clear line of communication between yourself and the customer. This is important as it ensures that the customer’s expectations are directly made known to you. Although the customer (typically a designer or manufacturer) may not understand the details of finite element analysis, they would have the most knowledge concerning design requirements and manufacturing issues. Understanding your customer’s needs is crucial to successful planning. It is a good idea to have a meeting to discuss the project before beginning to effectively plan or at least review the goals of the project.

Goals

1. Determine project goals, such as:

- Maximum deflections
- Peak stresses
- Stiffness under load
- Ultimate strength
- Fatigue life
- Vibration characteristics
- Thermal stresses
- Combinations of several requirements
- Optimization

2. Determine the specific objectives of the analysis.

- Form a consensus with all involved to establish objectives.
- Customers may be unable to clearly define goals.

If a clear plan is not agreed to in the beginning, a project will be continuously redirected.

History

Examine the components under investigation.

- Note the strengths and weaknesses of past performance.
- Examine the components in their working environment.
- Study reports written on similar studies.
- Discuss the problem with the authors of related reports, if possible.
- View films or operating tests to observe the components in action.

FEA Feasibility

Determine whether or not finite element analysis (FEA) is the best method to solve the problem. It may be necessary to consider other options such as classical analysis methods, especially if time is limited and the problem can be reduced to a simple calculation or handbook formula.

When modeling cannot be performed with existing software packages, the possibility of testing should be explored. Testing should always be considered if parts are available and a test method has been previously developed.

Whenever possible, the powerful combination of testing and finite element analysis should be employed. Test results provide useful insight into modeling and analysis while FEA results aid in the design and understanding of test procedures.

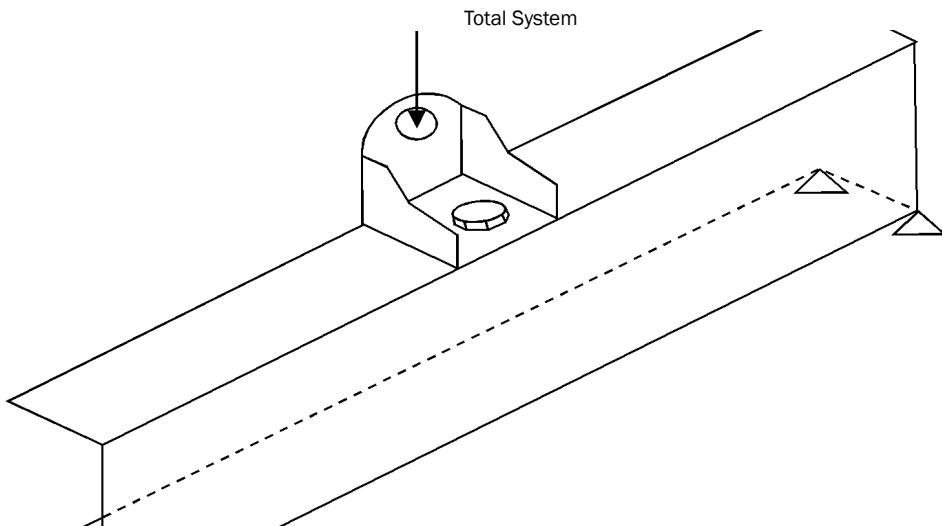
Time Constraints

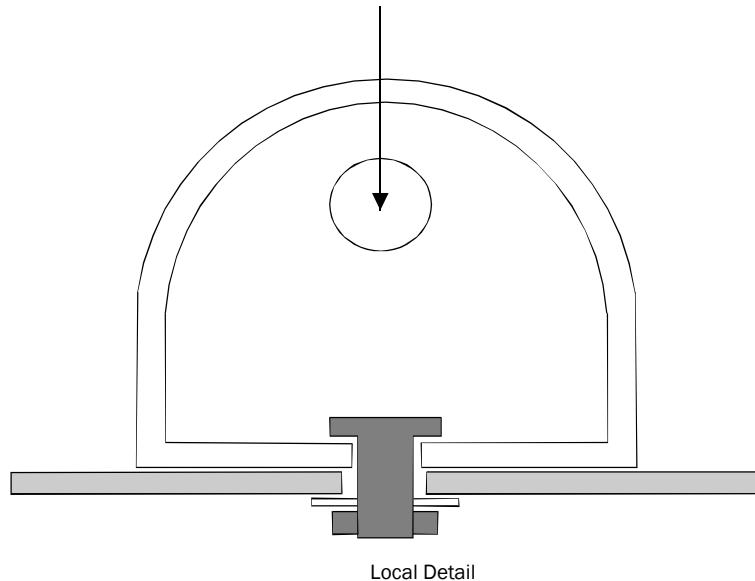
Know the timetable for conducting the analysis. This information will help to determine whether or not compromises in accuracy and confidence need to be made to deliver timely results. But before these types of compromises are made, concentrate your effort on reaching a compromise with the customer so that a more realistic amount of time can be allotted to the project. Consider the many details that need to be considered to obtain a solution, the effort involved in correcting results, and any miscellaneous problems that might arise.

Consider Boundaries And Surrounding Systems

Determine the amount of the surrounding structure that should be included in the model to obtain an accurate representation of the operating environment. Consider both local details and overall environment.

Consider the simple example below which has both localized and total structure issues. The bolted attachment alone could consume years of research if examined in detail, yet the support conditions may be more important to the overall performance of the channel section. Therefore, a determination must be made regarding the appropriate boundary conditions and loads. If historical data exists, it should be used to aid in this determination, and if used, be certain that you have a clear understanding of its relevance to the current project so that you can offer an intelligent response to questions regarding its use.





Decide whether the performance of the bracket, the rail, or the bolted connection is to be determined.

Plan Element Types And Model Size

Selecting element type(s) and target model size are important factors affecting timing, accuracy, and computational costs. Since there is usually **no single correct answer** to the question of how many and what type of elements to use, take the following points into consideration when making a determination.

- It is usually better to err on the “too detailed” side. Due to increasing speed and decreasing costs of computer cycles, fewer assumptions and higher accuracy can be achieved using this logic.
- Take care not to go to extremes because it will lead to extended turnaround times for model changes and increase time devoted to computer calculations. A good rule of thumb to follow is that the size of the problem grows with the square of the number of nodes in the model, and the solution time grows with the cube of the number of nodes. For example if the number of nodes doubles the memory required will be four times larger and the solution time will be eight times longer. In addition when the available RAM of the computer is exceeded then the solver must use a scratch file from disk and then the solution times are longer and unpredictable. Experience with a particular solver will help in future determinations of model size.
- Consider if the design information is detailed and complete. You may want to use less detailed models if you are evaluating several alternative concept designs. A completely developed design merits a more detailed finite element model.
- Solving subsequent coarse models will help you make decisions about element types and element density.

Review The Plan

“See the end before you begin” - conduct a review of the selected methods and modeling techniques. Keep the following questions in mind:

- What results will be most useful?
- What do you expect to happen?
- What things can go wrong?
- What questions will be asked when you present your results?

- Will you be prepared to answer questions the analysis has and has not addressed?
- How and in what form will you present your results?

You may choose to decide in advance which tables, x-y plots, stress plots, and deflection plots will support the questions that the analysis answers. Conducting these exercises will lessen the chance that an analysis will need to be re-run and will better prepare you to discuss your findings.

Planning The Solution Summary

Follow these steps to help ensure a successful analysis.

- Discuss the problem with all the appropriate people.
- Define the project goals as clearly as possible.
- Research the history of the component.
- Consider the application of alternative solutions.
- Be realistic in your objectives and approach based on the time available.
- Form a solution strategy based on the operating environment of the component.
- Select element types and number based on timing, accuracy, and computation cost.
- “See the end” before you begin in terms of final results requirements.

The 10 Questions You Should Be Able To Answer...

The following is a list of the 10 questions you should be able to answer about any given FE simulation. It is also a good checklist to bring along to presentations by others as a guide to assure all your questions are answered. If you have not or cannot answer these questions you need to go back and understand why.

1. What is the problem you are trying to solve?

- Problem statement
- Objective
- Constraints
- Design Variables (i.e. what is to be changed)

2. What geometry are you using?

- Do you trust the fidelity?
- Is the geometry changing?
- Can you change the geometry?

3. What solution type are you using?

- Is it consistent with your problem?

4. What is your material model?

5. What are your element selections?

6. What are your constraints?

- How are you holding the model?
- Over constrained or under constrained?

7. What are your loads?

8. How are you combining the loads and the constraints?

9. What are the results?

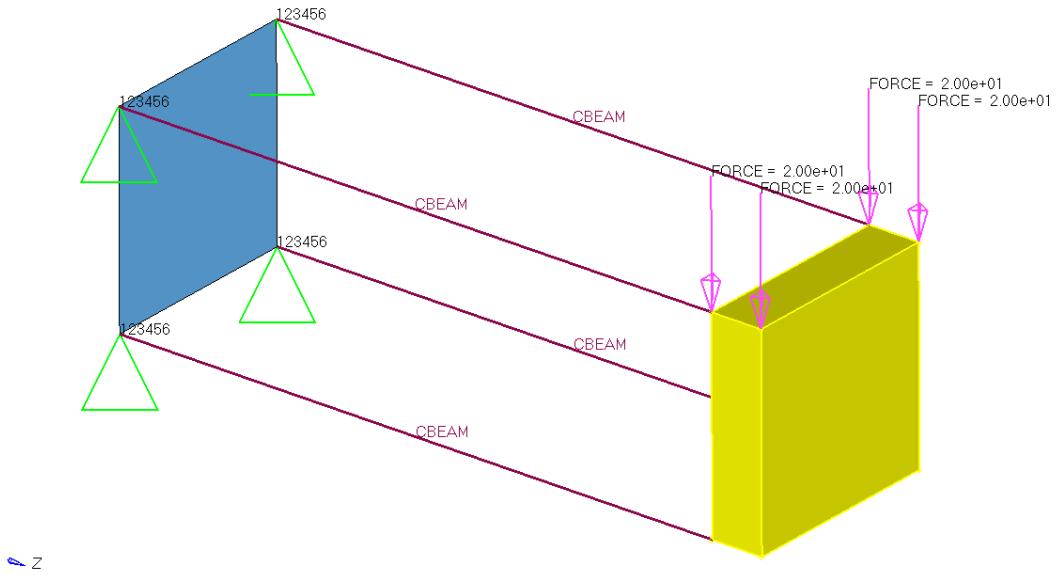
- What results are important for this problem?
- What is success?
- Are the results correct?
- How do you know?

10. What are your recommendations?

Sample Linear FEA Deck

Once the FEM modeling phase is completed, in other words the CAD geometry is meshed (i.e. approximated by finite elements), material and property information linked to the elements, loads and constraints assigned, the FEM model is exported from the pre-processor in a solver specific syntax (ASCII format).

An example of an OptiStruct solver input deck (ASCII format) is provided below.



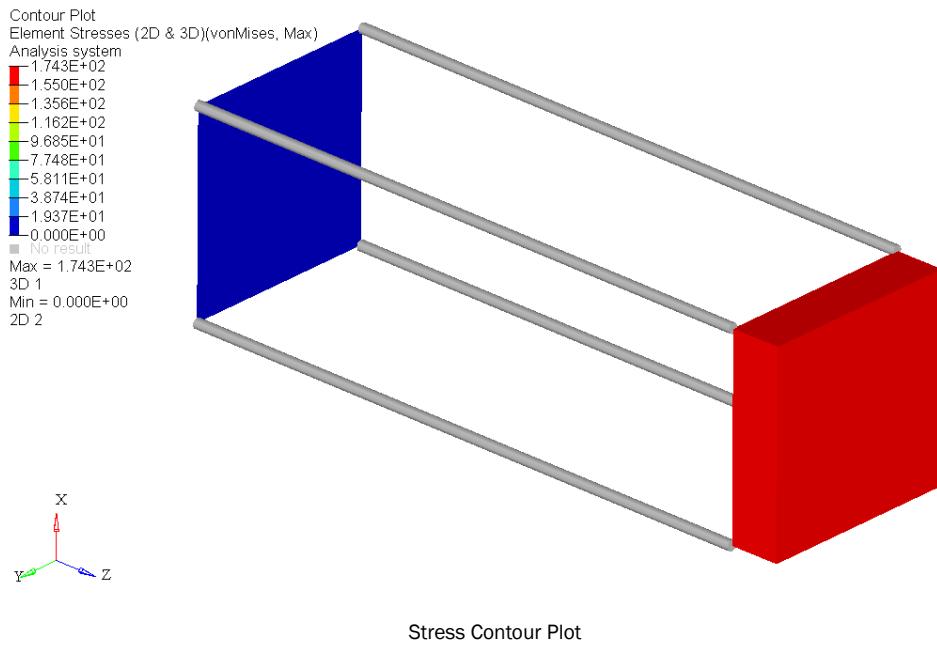
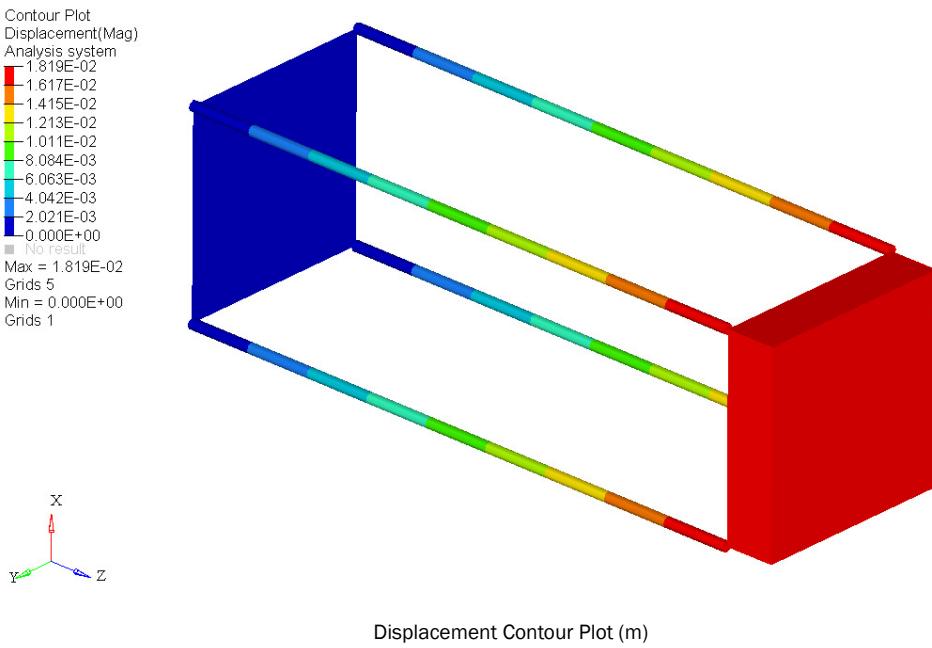
```

$$ Optistruct Input Deck Generated by HyperMesh Version : XXXX
$$ Generated using HyperMesh-Optistruct Template Version : XXXX
$$
$$      Case Control Cards           $ (Definition of the so-called load step / subcase)
$$-----$


$$
SUBCASE    1
SPC =      1
LOAD =     2
BEGIN BULK
$$
$$ GRID Data
$$
GRID    1    0.0   0.0   0.0
GRID    2    0.0   1.0   0.0
GRID    3    1.0   1.0   0.0
GRID    4    1.0   0.0   0.0
GRID    5    1.0   0.0   3.25
GRID    6    0.0   0.0   3.25
GRID    7    0.0   1.0   3.25
GRID    8    1.0   1.0   3.25
GRID    9    1.0   0.0   3.0
GRID   10    0.0   0.0   3.0
GRID   11    0.0   1.0   3.0
GRID   12    1.0   1.0   3.0
$$
$$
CBEAM    8    1    10    11.0   0.0   0.0
CBEAM    9    1    11    21.0   0.0   0.0
CBEAM   10    1    9    41.0   0.0   0.0
CBEAM   11    1    12    31.0   0.0   0.0
$$
$$
CQUAD4    2    2    4    3    2    1
$$
$$
CHEXA    1    3    5    6    7    8    9    10
+    11    12
$$
$$
PSHELL    2    10.25        11.0        10.8333330.0
$$
$$
$$
PSOLID    3    1
$$
$$
PBEAM    1    10.063  0.00033  0.00033  0.0    0.0
$$
$$
$$
MAT1    13.0+7      0.3
$$
$$
$$
SPC    1    4    1234560.0
SPC    1    3    1234560.0
SPC    1    2    1234560.0
SPC    1    1    1234560.0
$$
$$
FORCE   2    9    01.0   -20.0   0.0   0.0
FORCE   2    8    01.0   -20.0   0.0   0.0
FORCE   2    5    01.0   -20.0   0.0   0.0
FORCE   2    12   01.0   -20.0   0.0   0.0
ENDDATA

```

Sample Linear FEA Results



Accuracy Of The Results

In any FE analysis, the objective is to obtain some results that will help you solve some problem. In order to determine the success of the exercise you must have some feeling for the accuracy of your answers. The problem with this is that errors can be introduced in a FE analysis in many places.

Below is a list of some of them:

1. Solution type not matching the “**in vitro**” environment.
2. Material model or properties
3. Tolerances
4. Geometry considerations – detail captured
5. Elements
 - Shape- quality (Jacobian (Global – local))
 - Size – matching the detail of geometry
 - Type – matching displaced shapes, stress output
6. Loads and Constraints
7. Results
 - Not calculated where you want them
 - Interpretation
 - Averaging/Smoothing

Because of the many places that errors can be introduced into a FE analysis, it is sometimes a better practice to compare results that were achieved through comparable methods to see improvements in design. At least then the results are consistent. Consistency is really the key, because with consistency you can have predictability.

5.2 Creating A Solution Checklist

Use the following example of a solution checklist as a guide to develop your own checklist:

- Assign project number and describe the job to be completed.
- Set the goals of the solution. (This is done through communication with your customer).
- Identify resource and time constraints to the above goals.
- Evaluate past historical data.
- Describe the operating environment you are interested in simulating.
- Determine type of analysis that is applicable to the goals.
- Describe your selected boundary conditions.
- List the element types/combinations you have chosen.
- Determine your material model.
- List the assumptions you are making by selecting the analysis type, element selection, boundary conditions, and material model.
- List the limitations you expect as a result of the above assumptions. (Do the assumptions let you achieve your goal?)
- List the results you will present to attain the goal of the project.
- List the media by which you will present the results.
- Make a list of data required from outside sources.

5.3 Boundary Conditions And Load Cases

Study the boundary conditions (BCs) and applied loads of the component or system and establish proper sets before initiating finite element modeling and analysis. Keep in mind that even though chosen boundary conditions are an approximation of the actual physical conditions, they should match reality as closely as possible. Poorly chosen BCs can be misleading and produce costly design errors.

In some cases, several sets which bound the actual conditions may be appropriate. For example a flexible boundary may be considered partially fixed, completely fixed, or not fixed at all. The flexibility of a boundary should be included in the model if possible and the applied loads combined with the BCs should be in proper balance.

Methods which can help to understand and select BCs and load cases are explained in the following sections.

History

Research the history of similar systems by:

- Reading reports.
- Discussing similar projects with design engineers.
- Visiting test labs.
- Physically inspecting the components in their operating environment.
- Studying layout and detail drawings.

Check Environment

Determine the surrounding environment included in the study to capture the best approximation of the actual BCs. Is the area under study strongly or weakly coupled to its environment? This will be dependent on the types of attachments between different structures in the problem. Drawings, parts lists, and observation can tell you if two parts are welded, bolted, or pinned together. Each attachment scheme will have an impact on the assumed boundary condition. Creation of a free body diagram will greatly aid in the understanding of the boundary conditions.

Free Body Diagrams

Sketch free body diagrams of:

- Component and boundary forces.
- Attached and surrounding components.

Include all sets of forces in separate diagrams. Some of the forces will become BCs, others will become applied loads.

Gather Load Data

Gather known load information from previous tests and/or load simulations. You may need to initiate a test or simulation if no load data is available. Be certain that available data consists of a complete set (no missing values) since you cannot proceed without knowing the complete loading of the structure. If you do not know all the loading conditions, it is sometimes possible to assume "unit" loads that can be superimposed and scaled at a later time.

Minimize Number Of Boundary Condition Sets

Select the load and BC sets in a manner which will minimize the number of different BC sets. Minimizing the number of BC sets will reduce computer time since the number of matrix inversions that must be computed will be fewer.

Eliminate Rigid Body Motion

Be sure to provide at least enough boundary constraints to eliminate the possibility of rigid body motion. Rigid body motion occurs when the model is not fully constrained resulting in free motion in any of the six translation or rotation directions in the model. If this occurs, the solver translates this into a zero stiffness term in the global stiffness matrix and fails to solve the problem. If this requirement is not satisfied, the solver will either fail or give implausible results. (An exception to this rule is the solution for a free-free normal modes analysis.)

The inertia relief solution is a distinct and useful type of solution that requires the explicit identification of a minimum set of constraints. In this solution, the constraint forces are automatically eliminated by body forces with the results being similar to a free structure.

Remember that a finite element solution is an approximate solution and the assumption of a “balanced force system” may not be true to the solver. Even extremely small perturbations in a direction unconstrained will lead to rigid body motions.

Mechanisms

Mechanisms refer to the various mechanical connections that can be idealized with a set of boundary conditions. Below is a list of some common mechanisms and their corresponding boundary condition assumptions:

Support	ASSUMPTIONS	APPLICATION
Simple support	Reacts forces only in direction of support	Object resting on a support
Pinned support	Reacts forces but no rotations	Eyelet connection, clevis attachment
Fixed support	Reacts all forces and moments	Welds, bolted, riveted connection
Ball joint, Cable	Reacts only forces	Suspension ball joints
Pulley	Reacts forces at the center of the pulley and tangent to the cable connection.	Pulleys
Bushing	Reacts forces and moments perpendicular to the axis of rotation.	Automotive suspension bushing, roller bearings.

Symmetry

Symmetry can greatly simplify a problem by reducing its size and complexity both during the solution and preparation phases. Care must be taken however to ensure a consistent set of symmetry boundary conditions. **In order to use symmetry assumptions both the loads and boundary conditions and the actual geometry must be symmetric.** If either of these two criteria cannot be satisfied then the entire structure must be modeled. However, if the geometry is symmetric but the loads and boundary conditions are not, preparation of the model can still be simplified by using the symmetry assumption. Below is a list of common assumptions for setting up a plane of symmetry for both shells and solid elements. Constrain the degree of freedom that are shaded for the symmetry listed on the left. Note that beams and shells have the same assumptions. For more information on this topic please refer to Chapter 13 (Boundary Conditions and Loads).

	ux	uy	uz	rotx	roty	rotz
Shell xy						
Shell xz						
Shell yz						
Solid xy						
Solid xz						
Solid yz						

The plane of symmetry is indicated as xy, xz, yz, etc. respectively.

Superposition

Superposition is used to describe the practice of adding the force or load boundary conditions for a linear solution with the same constraints. If the constraints remain the same, the stiffness matrix only has to be solved once and, therefore, the loads can be applied separately. The results can then be added or superimposed during the results calculation stage. This can be useful in the practice of creating many “unit” loads and then both scaling and adding results together without rerunning the analysis. This technique requires that you consider as many possible unit loads as possible to avoid rerunning the analysis.

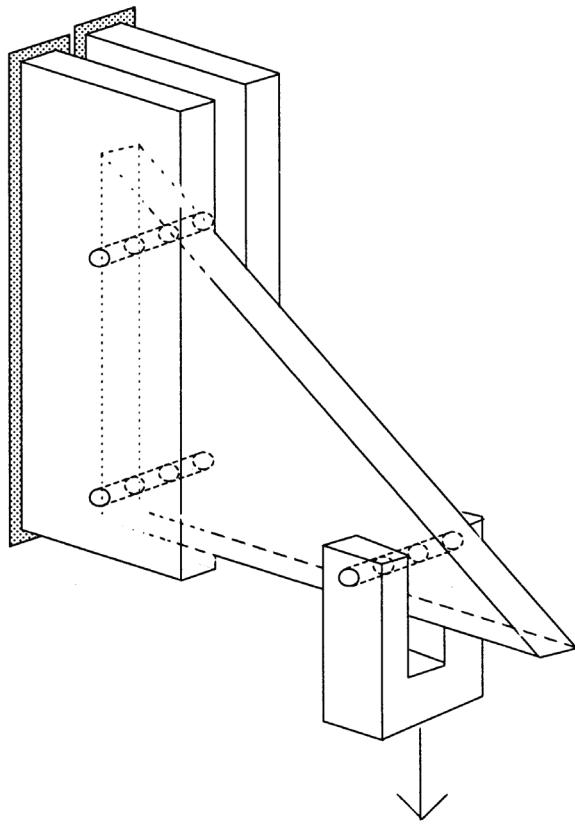
Choose Modeling Methods

Establish the modeling methods for load and boundary condition application. Determine whether point or pressure loads should be used, and whether the boundary conditions should be spread over an area or applied at a single point.

Keep in mind that errors always result at the point of application of loads and boundary conditions so choose a method to reduce the severity of the effects.

The following example illustrates the process of selecting modeling methods.

Clevis Support Bracket

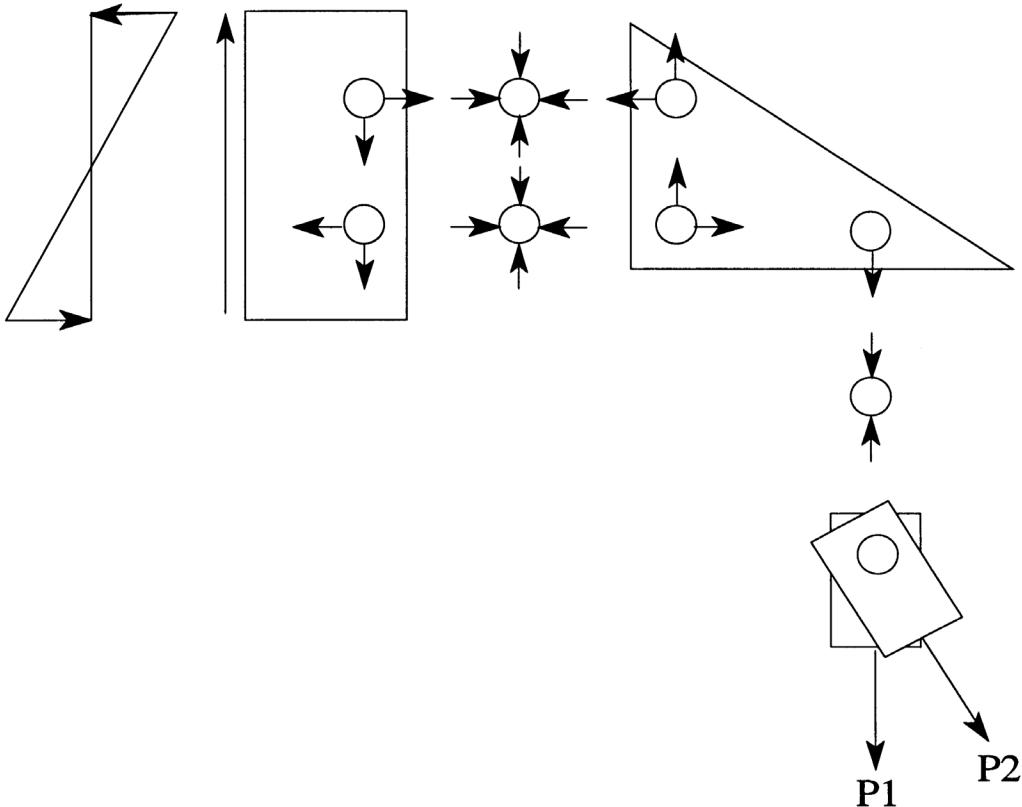


Clevis Support Bracket

The clevis support plate is of constant thickness and the loads are applied in the plane of the plate. Therefore, this is considered to be a plane stress problem. In a plane stress problem, all of the stresses are in the same plane as the applied loads. The plate is loosely coupled to the other parts by pins, therefore it can be analyzed separately with some confidence.

If the possibility that loads will occur out of the plane of the plate exists, a full 3D shell analysis should be considered. A plane stress element formulation could be used to solve this problem or, if some computation waste is acceptable, a shell formulation could be used. Shell elements will often provide more accurate results since their use is more universal and greater effort has been devoted to their development. The plane stress element formulation results in a significantly reduced stiffness matrix and therefore a faster solution time.

A full 3-D structural model with solid elements could be used if localized "through thickness" effects at the pins are to be considered. If interaction between the plate and the pin is to be investigated, a detailed and complex contact model would be required. A free body diagram of the system is shown in the following figure.



Free Body Diagram For Clevis Support Plate And Attached Structure

The above diagram only considers the effect of the P1 load case. The P2 load case would result in a horizontal component of force in the pin above the clevis and the plate at the attachment of the same pin.

Loads And Boundary Conditions Summary

1. Research the history of prior studies and conduct physical inspections.
2. Determine how much of the surrounding structure should be included.
3. Sketch and study free body diagrams of the component and attached structures.
4. Gather load information or conduct tests and or analyses to obtain loads.
5. Select load and boundary sets to study.
6. Check for a minimum number of constraints to eliminate rigid body motion.
7. Establish the modeling methods for application of load and boundary conditions.

5.4 Linear Assumption

Many problems in structural analysis can be solved with a linear finite element solver. The linear assumption arises due to the mathematics used to formulate the solutions. The physics of the problem can be greatly simplified if higher order terms are dropped from the deformations and the material behavior is assumed to be linear.

The following are a number of considerations that should be reviewed to determine if your problem can be solved using a linear assumption.

Small Deflections

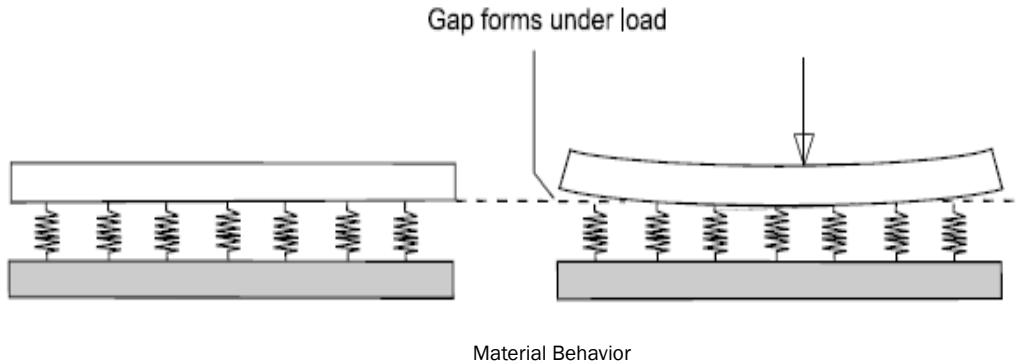
Determine whether the deflections obtained or predicted are small relative to the size of the structure. For thin structures, a deflection that is less than the thickness would be considered a small deflection. The deflection between two supports should be only a small percent of the distance between supports. This is especially true if the deflection causes a differential stiffness effect such as mid-plane stretching of a clamped plate.

Small Rotations

In linear codes all rotations are assumed to be small. Any angle measured in radians should be small enough that the tangent is approximately equal to the angle. Using this assumption, a ten degree angle introduces an error of approximately one percent in all related calculations. A thirty degree angle results in approximately a 10 percent error in deflection due to rotations assumed linear.

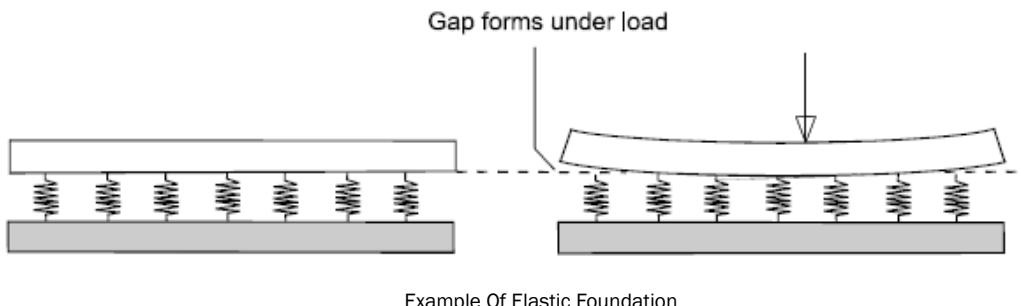
Material Properties

Linear solvers assume that all materials behave in a linear elastic manner. Some materials have a nonlinear elastic behavior, and although they do not necessarily yield, they still result in nonlinear structural behavior and require nonlinear codes for solution. If a structure is to be loaded beyond its yield point, nonlinear analysis would also be required. See the figure below for a comparison of material behavior.

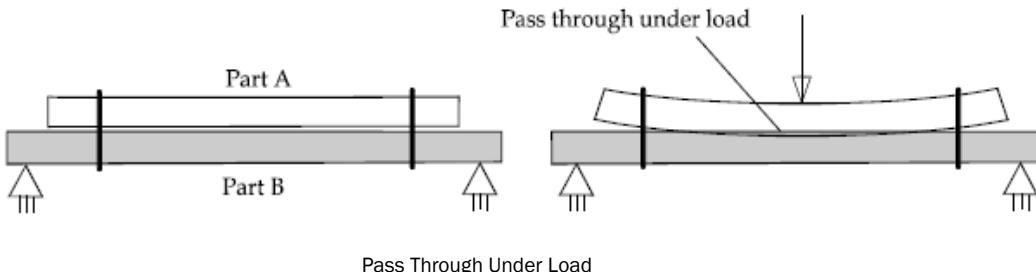


Constant Boundary Conditions

In order to correctly use a linear finite element program, the boundary conditions must not be dependent on the load application. The figure below illustrates an example where this is not true. A structure placed on an elastic foundation might tend to physically separate under the load, resulting in the formation of a gap. This gap is dependent on the load and therefore behaves as nonlinear.



Nonlinear contact is another source of nonlinear behavior illustrated in the figure below. Two parts are attached at several points and subjected to loading. The actual parts will come in contact and the load paths through the parts may be dramatically altered. In linear finite element analysis there may be a tendency for the parts to penetrate each other (sometimes called pass through). Since this is a load dependent nonlinear behavior, a linear solver will not handle this type of problem properly. Pass through can be handled in an approximate way by the addition of constraints between the parts normal to the contact surface. Care must be taken not to add too many constraints since separations may also occur. The constraint locations are also load case dependent.



Linear Assumption Summary

1. Deflections should be small relative to the structure.
2. Rotations should be less than 10 -15 degrees.
3. Material should be linear elastic.
4. Boundary conditions should be constant.

6 Common Mistakes And Errors

This chapter includes material from the book “**Practical Finite Element Analysis**”. It also has been reviewed and has additional material included by Matthias Goelke and Gareth Lee.

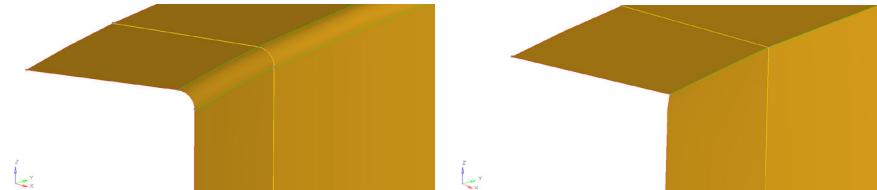
6.1 Modeling And Visualization

While the above summary reflects errors and mistakes from an organizational point of view, the following high level summary is about modeling and visualization errors. During the analysis, the FEM solver will report warnings and errors. While warnings can be considered as hints e.g. element quality is bad, errors cause the analysis to stop. Errors may be related to extremely distorted elements, missing material properties, rigid body modes due to insufficiently defined constraints,etc.

The modeling pitfalls listed below can be considered as “appetizers” with the intention of making you think (and worry) more about the model set-up. More in-depth details regarding the different modeling pitfalls are provided in the remaining chapters of this book.

Geometry Simplification

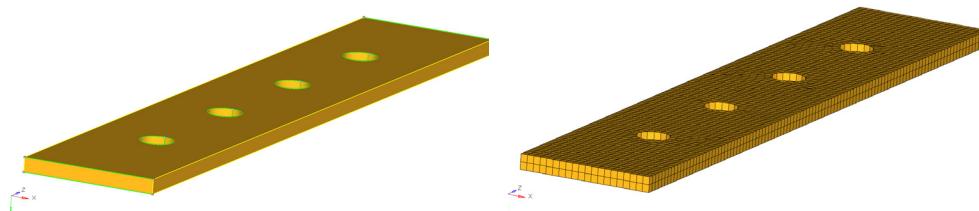
In many cases, it is appropriate or even required to simplify the imported geometry in order to achieve a better mesh quality. For instance, the required minimum element size must not be smaller than x millimeters. In order to solve this (project) related requirement, small fillets may be replaced by sharp edges, as shown in the images below. Even though this simplification was/is requested, keep in mind that your FEM model now “deviates” from the initial geometry.



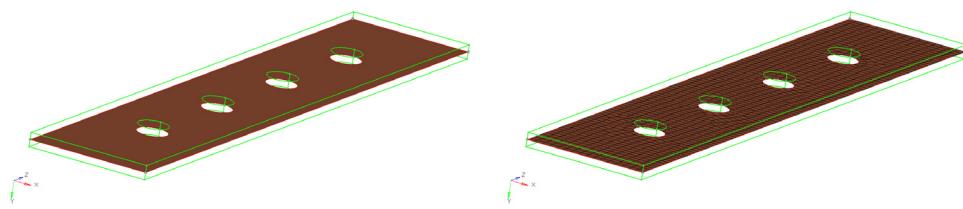
Meshing

What kind of elements are you using in your model? Why are you using this element type? Did you use this element type before?

You may mesh a thin walled 3D structure with 3D elements such as hexahedral or tetrahedral elements, or you may mesh the same structure with respect to its midsurface using 2D elements (trias or quads).



Model meshed with 3D elements



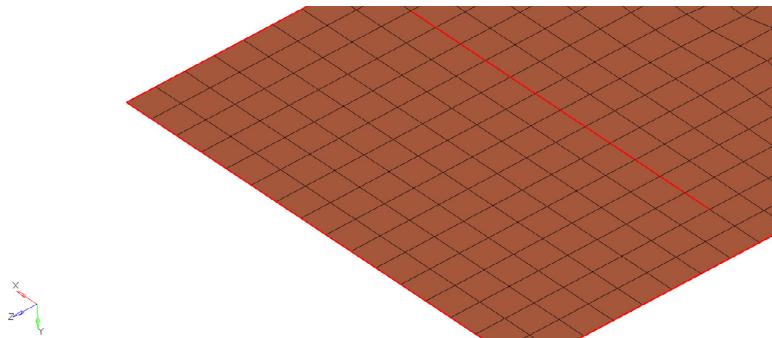
Model meshed with 2D elements

Aside from the “decision” of whether to use 2D or 3D elements, there are other “uncertainties” (or even errors) related to the different numerical characteristics of quad versus trias and hexahedral versus tetrahedral elements (see the chapters on 2D and 3D meshing).

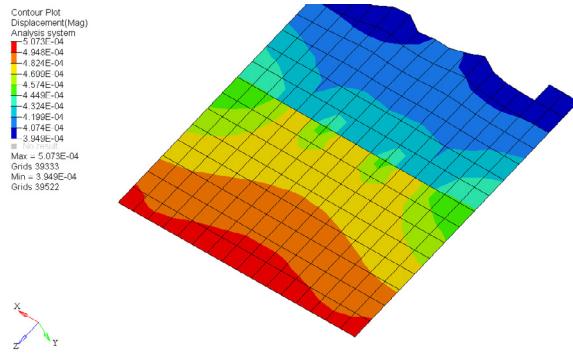
Another modeling error may be related to element size. The ultimate objective or aim is that the modeling results are independent of mesh size. Typically you need to re-run the analysis based on a finer mesh to check for convergence of the simulation results. As a rule of thumb, areas of interest should be meshed finer (smaller element size).

Of utmost importance is the element quality. Keep in mind that the elements not only “reflect” the CAD model, but eventually the analysis is based on the finite elements. Hence, any deviation from the ideal element shape (e.g. perfect quadrilateral shape in case of a quad element) introduces numerical errors. The magnitude of such errors is generally difficult to assess.

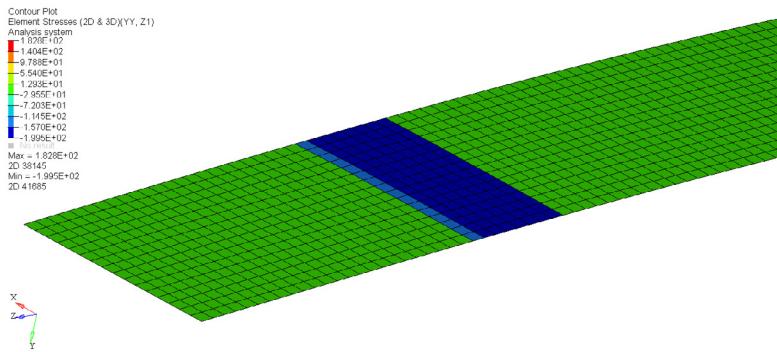
In the model shown below, some elements are not coupled to each other (i.e. duplicated nodes exist), hence the mesh is locally incompatible. The area along the edge where the elements are not coupled is marked in red.



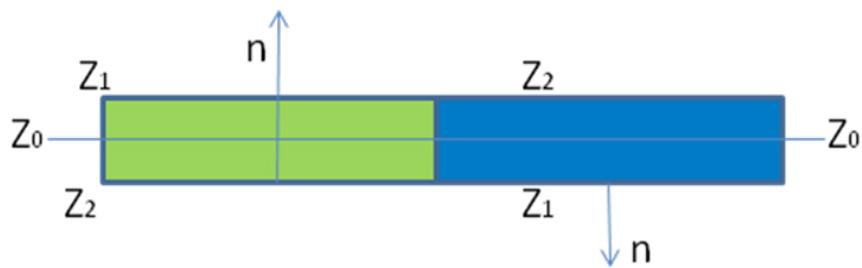
Still, the FEM program does not prompt any warning or error messages as this may be an intended model behavior. If the mesh is not intentionally detached (and the model is not checked for free edges), then this model error may remain unknown until the results are fully checked and understood. As shown in the contour plot below, the displacements are not continuous across some parts of the mesh.



Also, keep in mind the orientation of the element normals. In the image below, a simple plate subjected to bending is shown. The stress contour plot (at the base of the elements Z1) reveals a sudden change of its sign from bending (positive) to compression (negative).



The following figure helps to understand this situation. In the green area, Z1 is located at the top of the plate (tension) while in the blue area Z1 is located at the lower side of the element (compression).



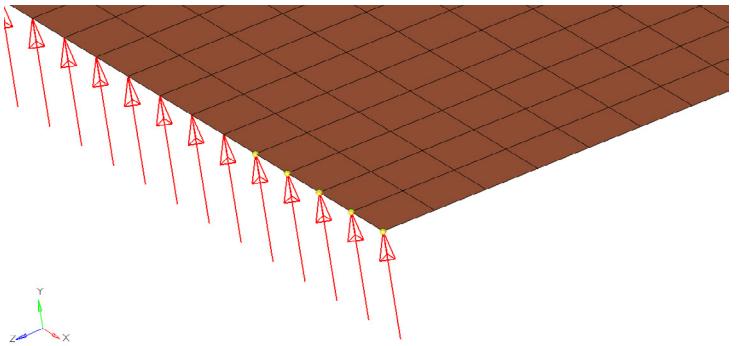
Material

Inconsistencies in your unit system represent another likely source of error, i.e. mixing millimeters with meters, kilograms with tons, etc. Be especially cautious if you need to convert properties from one system to another (e.g. pound-force lbf to Newton). There will be no warning message associated with any typos, except the “typo” will cause the entire model to “collapse” during analysis.

Boundary Conditions And Loads

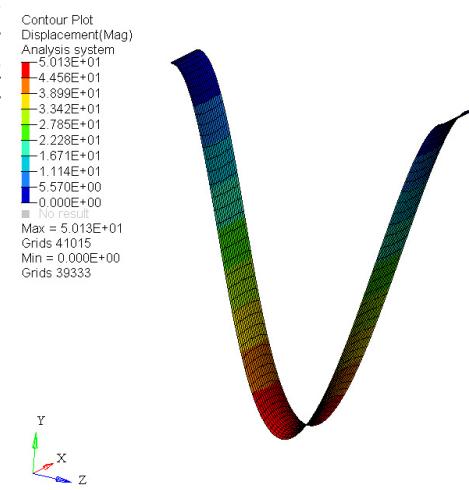
Applying boundary conditions and loads, as discussed in the chapter on Boundary Conditions and Loads, are extremely prone to errors. To be mentioned exemplarily, a modeling error may be introduced into the model by applying the constraints (or forces) to what is named temporary working nodes (in HyperMesh displayed as yellow nodes).

As the temporary nodes (yellow nodes in the image above) are not the same as finite element nodes, it may happen that the structure is not constrained or loaded as intended. “Ideally”, this may lead to rigid body modes (error message) or to questionable results due to an improperly constrained or loaded model.



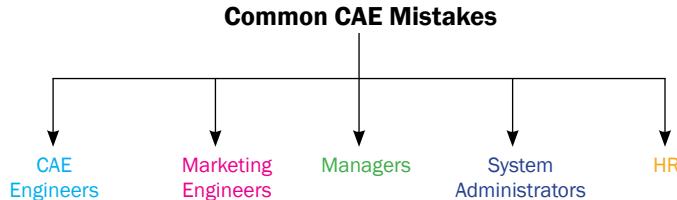
Visualization

When visualizing results, a false sense of achievement that the analyst might experience especially after having struggled with the model, could lower his/her attention regarding details while looking at the results. Quite often, especially while you are new to FEM, one becomes blinded by contour plots. Hence, always check the magnitude of displacements and stresses in the first step. Despite a reasonably looking displacements (or stress) contour plot, you may see displacement values in the order of 10^4 mm (small displacements assumed) or stresses far beyond 1000 MPa (linear elastic material).



6.2 Errors Within Organizations

Next, we will be sharing the mistakes made by CAE engineers working in different organizations.



CAE Engineers

1) Submission Of A Job Without Proper Cross Checking (Should Be Cross Checked By Ideally 2 CAE Engineers)

Checking the work of someone else is a rather boring and not willingly accepted job. But it is very important and a job of high responsibility. Checking and rechecking all the details ensures good quality and minimal mistakes.

For example, a CAE service-providing company submitted a meshing job to their regular client. Everything was perfect except for the material properties. The analyst at the customer end was used to error free models being submitted by this CAE team over the years. He blindly started the analysis without checking the material properties. At a later stage in the design process, a big difference was noticed in the results between the current analysis and the previous one carried out for a similar model. After checking both the models carefully, the analyst realized the difference in the material properties. Please be careful before submitting your work and check it several times and then ask your colleague to check it as well. Also, always request that your clients check the model in every aspect before starting the analysis.

2) Import / Export Errors

Some of the preprocessors do not export all the elements and boundary conditions that exist in the model, unless special options are turned on or special translators are used. For example, one CAE group exported a large mesh model with the template set for a specific external solver. Some of the special elements (RBE3 elements) were not exported due to a translator problem. These were extra rigid connections and did not result in rigid body modes during the free-free check. The analysis was carried out as expected by an analyst. Based on the CAE results, the CAD engineers released the drawings and a prototype was created. The test results were not satisfactory and further modifications were required. Updated CAD data was provided again to the same meshing group. The changes were local and it was to be carried out on the earlier submitted model. By this time, the CAE group had an upgraded version of the pre-processing software and the export operation was 100% successful (i.e. all the elements including the ones that were missed earlier were exported properly). The results for the modified model showed a drastic difference when compared to the original. After careful checking and comparing the number of elements, the analyst realized that a few rigid elements were missing in the first model. Hundreds of engineers had worked on the job in the mean time (CAD, prototype, testing, planning etc.). Who is responsible for this delay and cost? Is it the analyst, the service provider, or the pre-processing software? It's strongly recommended to import the mesh model before submission to the client (in a new file) and apply all of the quality checks as well as compare the number of elements of each type (such as number of tria, quad, rigid, spring, mass elements etc.).

3) Experienced Engineers Are The Best Guides And Teachers For Newcomers And Less Experienced Colleagues

CAE engineers are usually highly qualified (education wise) and having years of experience means that a lot of know-how and knowledge is available within the team. The best teachers for newcomers in any organization are undoubtedly the senior engineers working in the same group. Software trainers or consultants do not know exactly what is required by the customer. Every company should encourage and pay special incentives for experienced engineers to share their knowledge with newcomers.

4) Meshing Is Considered Low Level Work, Post Graduates And PhD's Are Reluctant To Spend Time On Meshing

Sometimes a dangerous trend is observed among post graduates and PhDs. They feel meshing is a low level job and being highly qualified, they should not waste time in such low level work. A building cannot be built on a weak foundation. Meshing is the foundation of CAE. At least in the initial years, analysts should be encouraged to mesh the components.

5) CAE Engineers Are Reluctant To Visit The Shop Floor, Testing Department, Or Field To Study The Manufacturing, Functioning, And Failures Of The Components

Just sitting in front of the computer in an air-conditioned office and submitting nice analysis reports is not going to make the analysis successful. What is absolutely necessary is to regularly visit the test department, observing the components on the structure, and comparing the real life performance with results of the computer model. These days, many times a CAE team is located in a different country than the manufacturing and testing facility. The quality of the CAE work would be much better if there is an opportunity to know the product, manufacturing process, testing and in-field behavior.

6) Providing Basic Training Related To Data Acquisition And Testing

At least a basic training on the data acquisition and testing methods is strongly recommended for CAE engineers.

7) Unnecessary Emphasis On Modeling The Minute Details Without Giving Due Consideration To Available Time, Hardware, And Software Capabilities

Finite Element Analysis is an approximate approach. Modeling the things to the minute details without giving due consideration to the capabilities of the available software and hardware could unnecessarily complicate the problem. For example, when analyzing a structure and the failure is expected at the body, a bolt should not be simulated by modeling the minute details like threads. Instead a beam element and connections using rigid in the washer area could adequately model the bolt (in the linear static domain).

8) Loyal To Specific Software And A Resistance To Learn And Use New Ones

Engineers using a specific software for years, are not willing to switch to other ones. No commercial software is perfect and every software has its own advantages and disadvantages. In the service industry, what matters most is the time and quality of the work. If a specific software is good but takes more time in comparison to another one for some specific application, then it is better to use the better one. Sometimes a combination of two different software works faster. For example, meshing in one software and then performing quality improvement or remeshing in other. A CAE engineer should be loyal to his/her duty rather than a specific software.

9) Not The CAE Engineers But The Design Engineers Are The Most Important Person In The Design Chain

CAE engineers are usually highly qualified, paid higher salaries and sometimes it leads to a superiority complex (that they are the most important people in the design cycle process). But it should always be remembered that the Design engineer is the most important person and the role of CAE engineers is to provide analysis services to him/her (other service providers are test, purchase, manufacturing, etc.).

10) While Suggesting The Modifications, No Consideration Is Taken For The Manufacturing Constraints And The Cost Effectiveness

Sometimes the CAE engineer gives suggestions which are either not manufacturable or cost effective. For example, it is very easy to increase the thickness of the parts showing a higher stress, or to suggest that high strength (costly) material should be used, or to suggest geometry modifications without considering the manufacturing constraints. Sometimes CAE engineers are adamant about their proposals and are not willing to carry out further iterations as per suggestions from the design or manufacturing engineer.

CAE Marketing Engineers

1) Accepting Jobs Beyond Their Capabilities

Sometimes marketing engineers accept jobs just because it is from a reputed company or because the volume of work is very large, without giving due consideration to the capabilities and limitations of their technical team and available software / hardware.

2) Promising Unrealistic Time Schedule:

Sometimes marketing engineers promise to deliver results in a time span that is not possible with the current strength of the team and the number of software seats available. Maintaining a strict time schedule with good quality work is necessary and reflects

the successful marketing of any company. Sweet talking, impressive infrastructure, and other facilities can create an excellent first impression with the client during initial visits, but it will vanish in no time if the delivery schedule isn't maintained and the quality of the work is poor.

CAE Managers And Group Leaders

1) Committing To The Job Without Consulting The Cae Engineer

In particular, during a visit to the clients, managers whose domain expertise is not CAE are involved in the meetings and sometimes commit to the job without consulting the responsible person.

2) CAE Manager / Group Leader Should Be Someone Who Has Spent Several Years In The Field, Not The One Who Is An Expert In Another Area Or A Non-Technical Manager

A CAE experienced manager understands the problems faced by CAE engineers and is capable of helping them personally when required. A CAE job is supposed to be a white collar job, but it really requires day and night hard work and involves considerable mental stress due to tight delivery schedules. The following replies from managers or group leaders could be very frustrating and demoralize any CAE engineer "don't ask me, that's your job" or "you should know these simple things" or "if you cannot do it, we will find someone else" or "I want results and not the problems, don't come to my office without the results, understood!".

CAE Process Management

CAE Process Management is helping organizations to capture knowledge and ensure best practices for CAE. It can be used for CAE load case automation, process guidance, and process integration. It lets organizations implement standardized CAE processes that automate the load case setup, interface with CAD, PDM systems, databases, and other IT systems and applications by capturing the "best practices" as templates. It also helps them retain knowledge, even when people quit and leave organizations, besides improving productivity and reliability of doing CAE between differently skilled CAE users.

CAE System Administrators

1) Laziness In Fixing Computer Hardware / Software Related Problems

In a group of 15 CAE engineers, a minimum of 15 workstations are required. Computers are, after all, machines and bound to create problems. It will not be logical to expect the best performance from the team without providing them good computers and fixing the hardware, software problems immediately. A knowledgeable and prompt system administrator is a very valuable asset for any CAE group.

2) Improper Data Backup Process

One cannot afford data loss at the midpoint or at the project completion phase. Imagine what the impression of the organization would be if they say to the client that although the job was almost finished, the system crashed and we did not have proper data backup. It is the responsibility of a system administrator and CAE manager to ensure the simple arrangement for the daily backup and forcing all the users to backup their data before leaving the office. Another good practice is to avoid working in a single file throughout the process. Instead, save the file with a different name after every 3 or 4 hours of work or whenever deemed appropriate, is recommended.

Human Resources (HR)

Inconsistent Salaries For The Same Post And Same Job Profile

In many (probably all) organizations, the most infamous department is HR. In CAE groups, it is common to find inconsistent salaries for the same qualification and the same job profile. When there is an urgent requirement, HR people generally offer higher salaries. When engineers come to know that the newly recruited person has been offered much more than what they are getting, they feel frustrated. Sometimes, this results in HR recruiting one new engineer and as a result, two existing engineers leave.

7 Consistent Units

This chapter includes material from the book “**Practical Finite Element Analysis**”. It also has been reviewed and has additional material included by Gareth Lee and Sascha Beuermann.

Numerous errors in engineering calculations can be attributed to the careless use of units. To avoid errors, follow these basic rules:

- Always use a consistent set of units for each problem.
- Develop a sense of the physical reality of the units being used and think in terms of those units.
- Always apply the test of reasonableness to input and output quantities. If a number appears to be out of the expected range, it probably is.
- Be particularly careful when using someone else’s work. They may have used a conversion factor but failed to clearly document it.
- Avoid conversion factors and non-standard units.
- Most general purpose analysis codes have no built-in knowledge of units, making unit consistency the analyst’s responsibility.

The fundamental units of measure used in structural analysis are:

- Mass
- Length
- Time
- Temperature

All other units are based on these fundamental units. Nowadays the International System of Units (SI) is used generally to mark the unit of dimension.

A consistent set of base and derived units, commonly used in engineering, is given in the table at the end of this chapter.

7.1 Equations Used To Help Determine Consistent Units

To test whether a set of units is consistent, one should check basic equations. For example, in the case of force,

$$F = m \cdot a = m \cdot l / t^2 \text{ and therefore the definition of a Newton is } 1 \text{ N} = 1 \text{ kg} \cdot 1 \text{ m/s}^2$$

Please see the Chapter about “Material and Property” Information, for examples of sets of consistent units.

Be aware that most codes use angular measures in degrees in some input and output. For instance, RADIOSS uses degrees when inputting angles in cylindrical or spherical coordinate systems but outputs displacement angles in radians. Refer to the appropriate users manual to determine which measure to use for angles.

Physical constants are also closely related to unit systems. Be certain that the constants you look up or “inherit” from others are in the same consistent unit set that you have chosen. Do not blindly accept constants passed on to you from another person. It is best to look up constant values in a recognized source and reference the source in your documentation.

Complex analytical projects involve a great number of details, each of which must be correct in order to obtain an acceptable solution. Seek assistance if you do not feel comfortable with the magnitude of detail checking required or if you are not confident about your intuition regarding the “typical” answers that your type of problem should produce. To increase your skills and confidence, solve a scaled down version of the problem you are working on or, solve problems that have a handbook or closed form solution.

Quantity	Symbol	Dimension	SI-System (MKS)	System mm-t-s Unit	Mult.	System mm-kg-ms Unit	Mult.
Length	l	L	m	mm	10^3	mm	10^3
Mass	m	M	kg	t (tonne)	10^{-3}	kg	1
Time	t	T	s	s	1	ms	10^3
Temperature	T	degrees	K	K	1	K	1
Work, Energy	W, E	ML^2T^{-2}	$J=Nm=W \cdot s$	mJ	10^3	J	1
Acceleration	a	LT^{-2}	$m \cdot s^{-2}$	$mm \cdot s^{-2}$	10^3	$mm \cdot ms^{-2}$	10^{-3}
Area	A	L^2	m^2	mm^2	10^6	mm^2	10^6
Frequency	f	T^{-1}	$Hz=s^{-1}$	$Hz=s^{-1}$	1	ms^{-1}	10^{-3}
Velocity	v	LT^{-1}	$m \cdot s^{-2}$	$mm \cdot s^{-1}$	10^3	$mm \cdot ms^{-1}$	1
Volume	V	L^3	m^3	mm^3	10^9	mm^3	10^9
Angular Acceleration	a	T^{-2}	$rad \cdot s^{-2}=s^2$	$rad \cdot s^{-2}=s^2$	1	$rad \cdot ms^{-2}=ms^2$	10^{-6}
Angular Velocity	w	T^{-1}	$rad \cdot s^{-1}=s^1$	$rad \cdot s^{-1}=s^1$	1	$rad \cdot ms^{-1}=ms^1$	10^{-3}
Density	r	ML^{-3}	$kg \cdot m^{-3}$	$t \cdot mm^{-3}$	10^{-12}	$kg \cdot mm^3$	10^{-9}
Pressure, Stress, Young's Modulus	p, s, t, E	$ML^{-1}T^{-2}$	$Pa=N \cdot m^{-2}$	$MPa=N \cdot mm^{-2}$	10^{-6}	$GPa=kN \cdot mm^2$	10^{-9}
Force	F	MLT^{-2}	$N=kg \cdot m \cdot s^{-2}$	N	1	kN	10^{-3}
Moment	M	ML^2T^{-2}	$N \cdot m$	$N \cdot mm$	10^3	$kN \cdot mm$	1
Stiffness	c	MT^{-2}	$N \cdot m^{-1}$	$N \cdot mm^{-1}$	10^{-3}	$kN \cdot mm^{-1}$	10^{-6}

The given conversion multipliers (Mult.) are valid for conversion from the SI-System to the given system.

Consistent Unit Summary

- Use a consistent set of units which you fully understand.
- Know your sources of physical constants.
- Be careful with decimal points.
- Study problems with known answers to gain confidence.
- Solve scaled down versions of the larger problems you are working on.

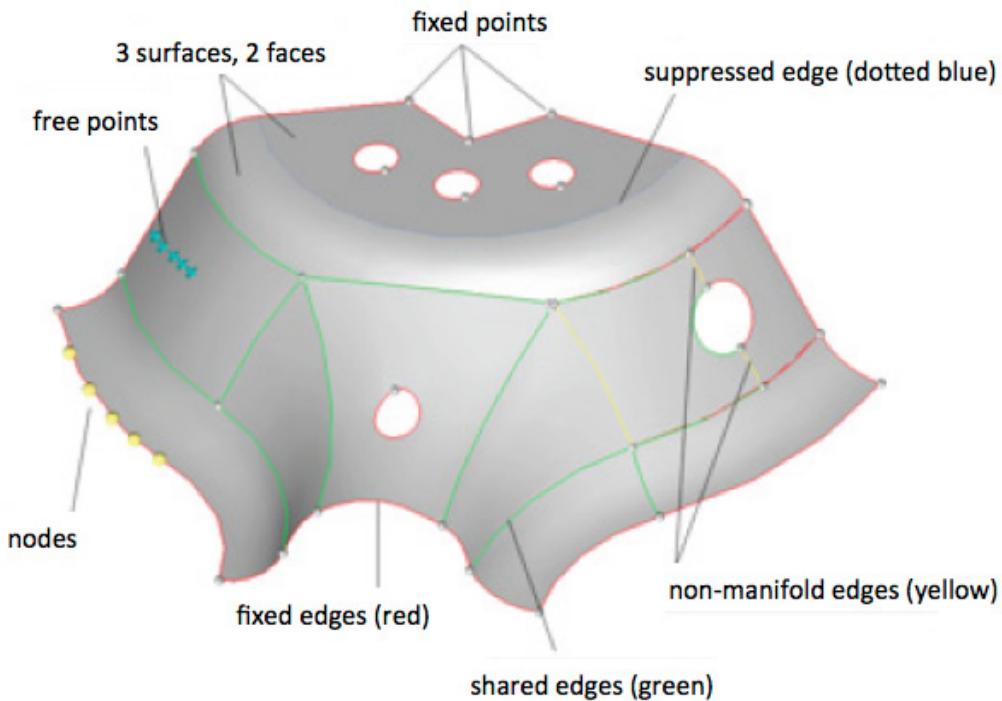
8

Geometry In HyperMesh

This chapter has been reviewed and has additional material included by Rahul Ponginan, Prakash Pagadala and Matthias Goelke.

8.1 HyperMesh Geometry Terminology

While dealing with geometry it is important to be familiar with the relevant HyperMesh terminology:
As mentioned in the previous chapters, your CAE project typically starts with the import of given CAD data e.g. Catia, STEP, UG, IGES, SolidWorks, solidThinking etc. (of course, you may create your CAD model in HyperMesh as well).

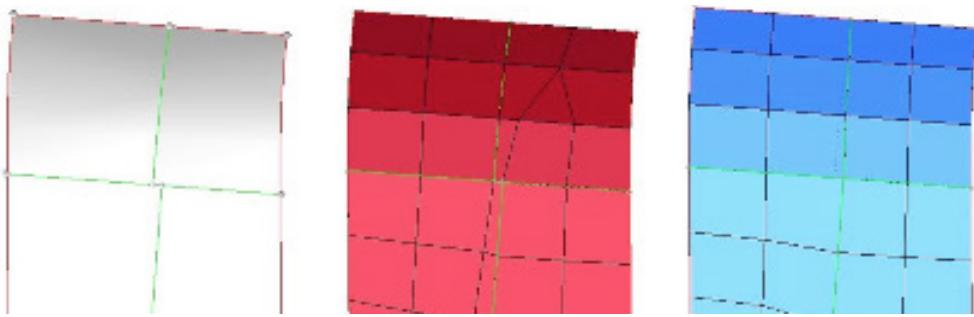


While the importation of data generally occurs with little error, there are issues that can occur, and as such, HyperMesh offers a wide variety of tools to remedy these geometric issues (this is one of the many reasons why HyperMesh is used in so many places).

Some of the issues described below do exist because when designers create CAD geometry, their priorities are different from those of analysts trying to use the data. For a designer, a single smooth surface is typically split into smaller patches.

Some resulting geometry issues:

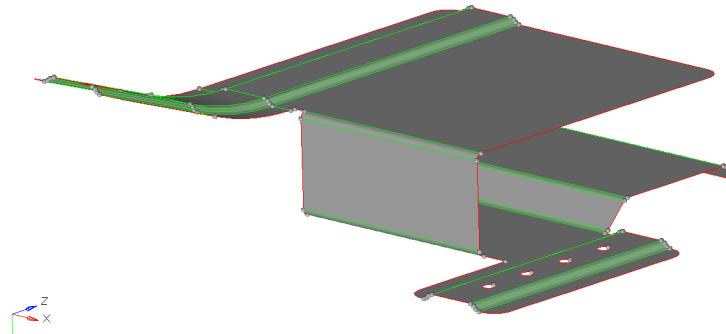
- Surfaces are not stitched together (i.e. there is a gap between surfaces)
- Very small surfaces are squeezed between regular surfaces
- The juncture between two surfaces often contains gaps, overlaps, or other misalignments



Left: Cad With “Jumps” Results In Irregularly Shaped Elements (Middle). Right: Improved Cad With Regular Mesh

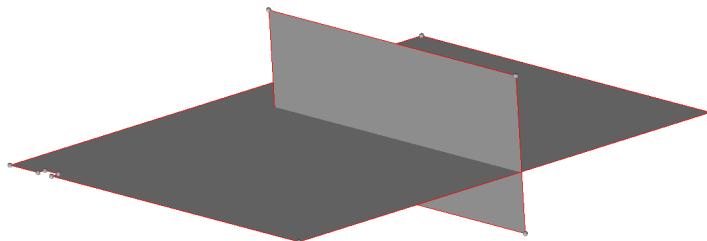
- The geometry is a thin-walled volume structure, i.e. instead of building a complex and exhaustive 3D mesh, a corresponding mid-surface model meshed with 2D elements would be far better

Original Thin Walled Structure

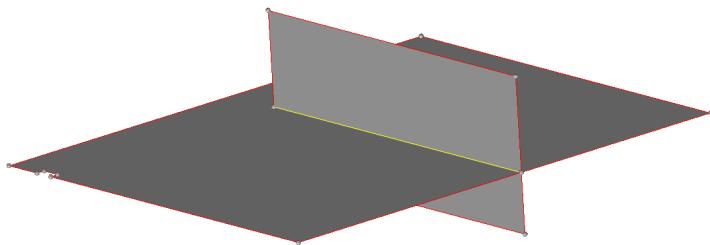


Midsurface Representation Of The Thin Walled Structure

- Surfaces penetrate each other (such as at a t-connection) but don't “feel” each other

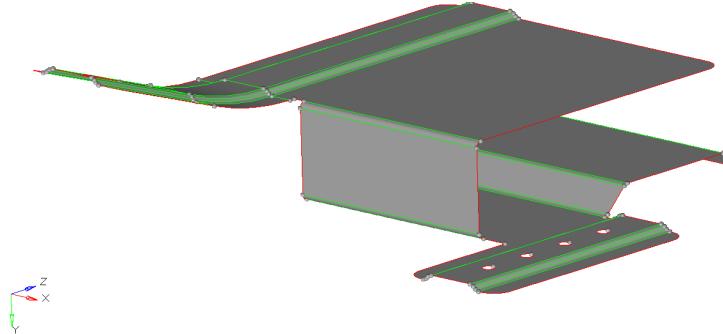


Original Geometry

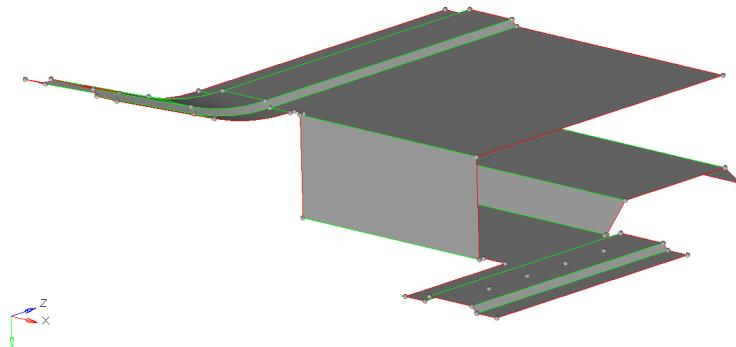


Geometry Fixed So That There Is Proper Connectivity

- Geometry is much too detailed (e.g. tiny fillets which are not needed for the analysis)



Original geometry, note the small fillets



Simplified Geometry Where Small Fillets Are Replaced By A Sharp Edge

- And many others ...

8.2 Geometry Cleanup

All of the issues defined in the previous section typically demand what is called **Geometry Cleanup**.

Topology Repair: Strategy

Below is a general strategy that can be followed to perform the topology repair. This is a generalized strategy which may need to be changed to suit the needs of your model, but it provides a good starting point to perform the topology repair.

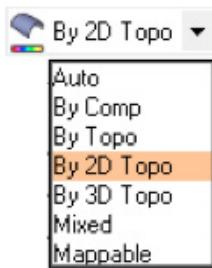
1. Understand the size and scale of the model

- With models that represent everything from full size ships to microscopic electronic parts all residing in a graphics area on a computer monitor, it is often difficult to understand the overall scope of the model. It is critical to get an idea of the overall size of the model and determine a global element size that will be applied to the eventual mesh.

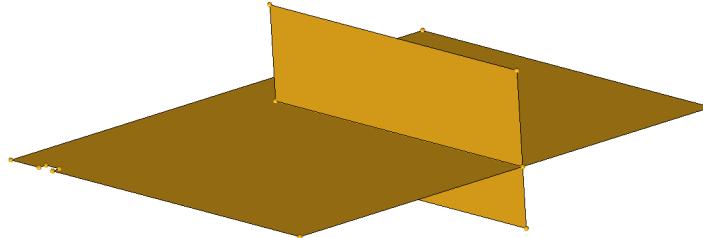
2. Set a cleanup tolerance based upon the previously determined global element size.

- With the element size established, a cleanup tolerance can now be set. The cleanup tolerance specifies the largest gap size to be closed by the topology functions. This value should never exceed 15-20% of the global element size. Values beyond this limit can introduce distortion into the mesh.

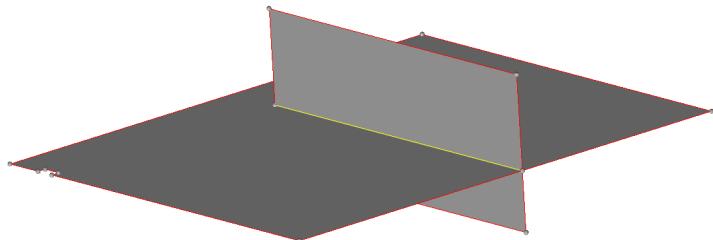
3. Use topology display tools to determine what needs to be fixed. For instance, to display the topology of 2D geometry set the selector to By 2D Topo



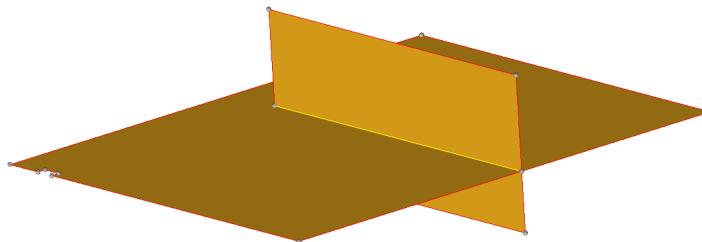
- Visualization mode: **By Comp** (which assigns the color of the component)



- Visualization mode: **By 2D Topo**



- Visualization mode: **Mixed** which assigns the component color and adds topology information



4. Find duplicate surfaces and delete them.

- To delete duplicate surfaces, from the menu bar select Geometry > Defeature > Duplicates.

5. Use equivalence to combine as many free edge pairs as possible.

- Visually verify no surfaces were collapsed with this function.

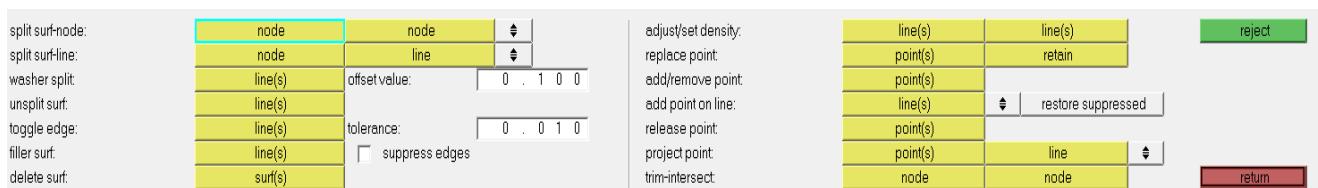
6. Use toggle to combine any remaining edges.

- Use replace if more control is needed.

7. Use filler surface to fill in any missing surfaces.

8. The equivalence, toggle, and filler surface can be accessed within the Quick Edit panel.

- To access the Quick Edit panel, from the menu bar select Geometry > Quick Edit.



Topology Repair: Tools And Panels

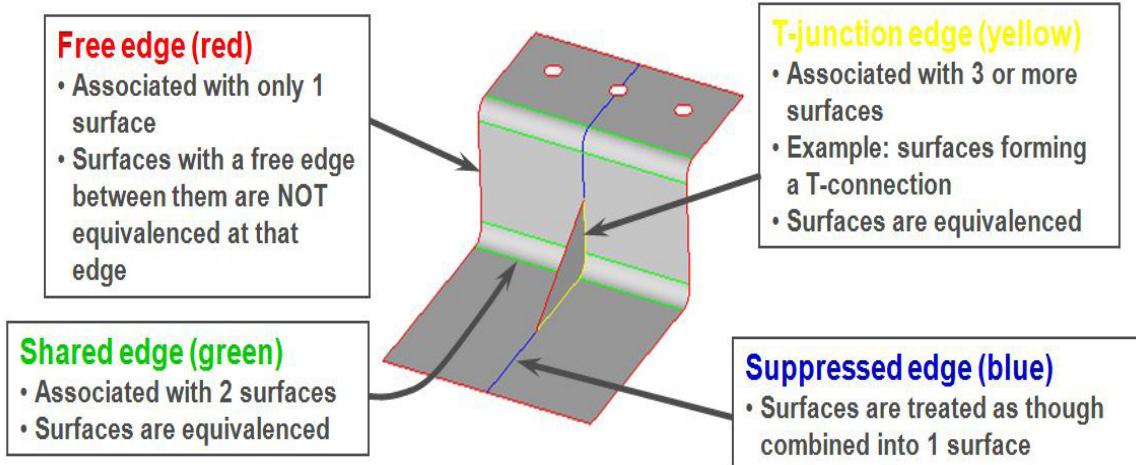
The perimeter of a surface is defined by edges. There are four types of surface edges:

- Free edges
- Shared edges
- Suppressed edges
- Non-manifold edges

Surface edges are different from lines and are sometimes handled differently for certain HyperMesh operations. The connectivity of surface edges constitutes the geometric topology. Below the four types of surface edges which represents the geometric topology is described (Note: the shown model is displayed via the **2D Topo** mode in HyperMesh).

Free Edges

A free edge is an edge that is owned by only one surface. Free edges are colored red by default.



On a clean 2D model consisting of surfaces, free edges appear only along the outer perimeter of the part and around any interior holes.

Note: Free edges that appear between two adjacent surfaces indicate the existence of a gap between the two surfaces. The automesher will leave a gap in the mesh wherever there is a gap between two surfaces.

Shared Edges

A shared edge is an edge that is owned, or shared, by two adjacent surfaces. Shared edges are colored green by default.

When the edge between two surfaces is a shared edge (this is what you typically want to have), there is no gap or overlap between the two surfaces - they are geometrically continuous. The automesher always places seed nodes along the length of a shared edge and will produce a continuous mesh without any gaps along that edge. The automesher will not construct any individual elements that cross over a shared edge.

Suppressed Edges

A suppressed edge is shared by two surfaces but it is ignored by the automesher. Suppressed edges are colored blue by default.

Like a shared edge, a suppressed edge indicates geometric continuity between two surfaces but, unlike a shared edge, the automesher will mesh across a suppressed edge as if it were not even there. The automesher does not place seed nodes along the length of a suppressed edge and, consequently, individual elements will span across it. By suppressing undesirable edges you are effectively combining surfaces into larger logical meshable regions.

Non-Manifold Edges

A non-manifold edge is owned by three or more surfaces. Non-manifold edges are colored yellow by default.

They typically occur at "T" intersections between surfaces or when 2 or more duplicate surfaces exist. The automesher always places seed nodes along their length and will produce a continuous mesh without any gaps along that edge. The automesher will not construct any individual elements that cross over a T-joint edge. These edges cannot be suppressed.

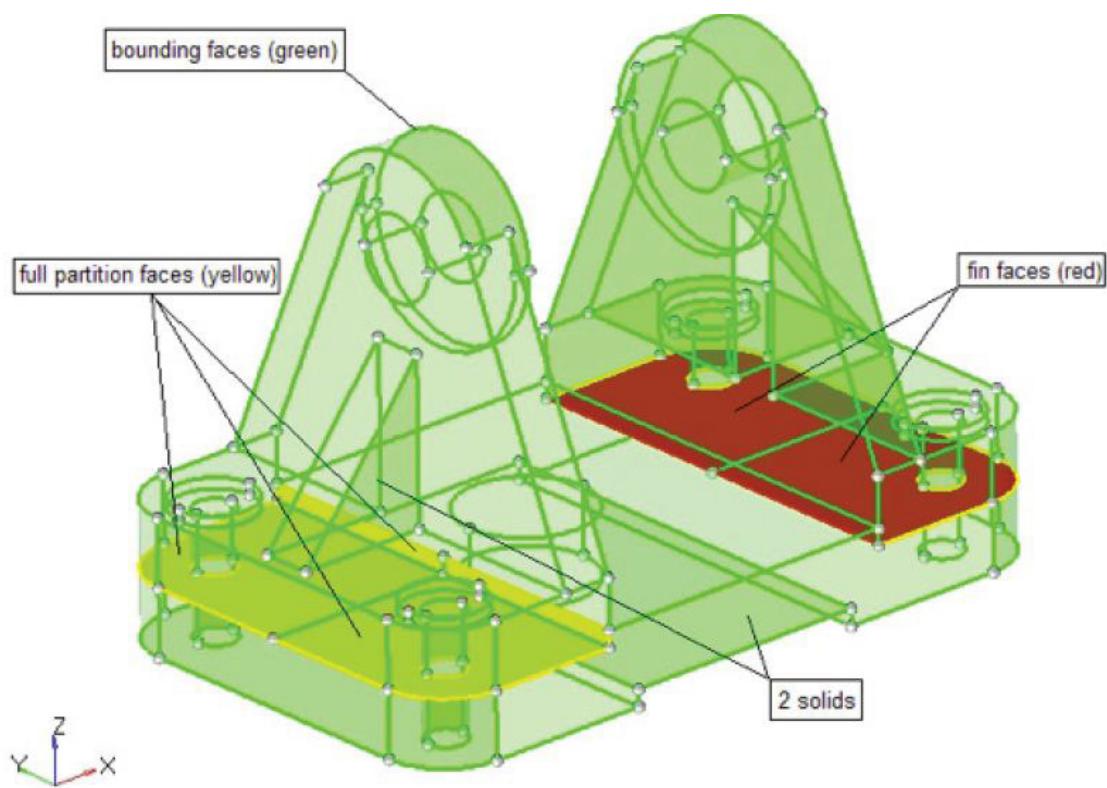
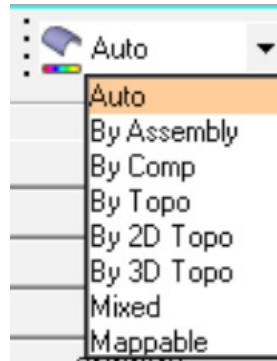
Solids

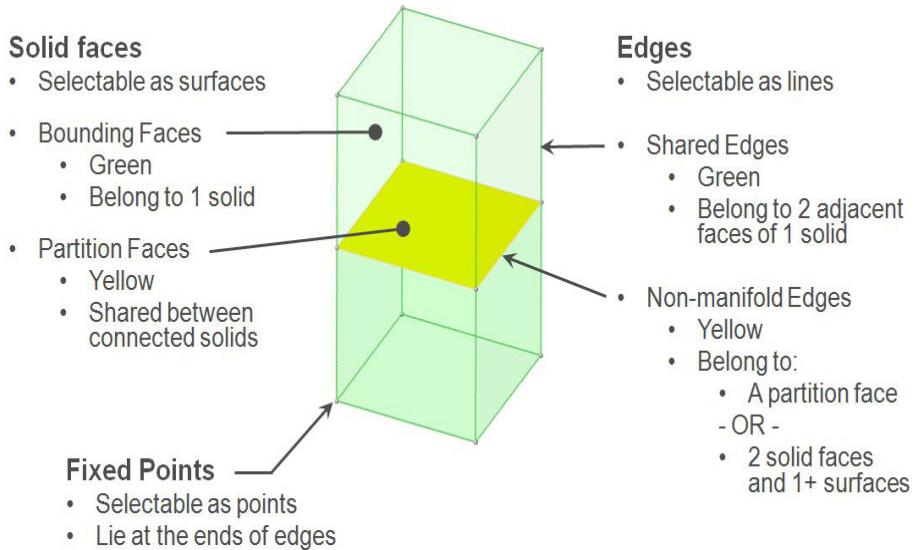
A solid is a closed volume of surfaces that can take any shape. Solids are three-dimensional entities that can be used in automatic tetra and solid meshing. Its color is determined by the component collector to which it belongs.

The surfaces defining a solid can belong to multiple component collectors. The display of a solid and its bounding surfaces are controlled only by the component collector to which the solid belongs.

Below is an image of solid topology as well as a description of the three types of surfaces which define the topology of a solid.

To activate the 3D topology mode view, please activate the corresponding setting





Bounding Faces

A bounding face is a surface that defines the outer boundary of a single solid. Bounding faces are shaded green by default.

A bounding face is unique and is not shared with any other solid. A single solid volume is defined entirely by bounding faces.

Fin Faces

A fin face is a surface that has the same solid on all sides i.e. it acts as a fin inside of a single solid. Fin faces are shaded red by default.

A fin face can be created when manually merging solids or when creating solids with internal fin surfaces.

Full Partition Faces

A full partition face is a surface that defines a shared boundary between one or more solids. Full partition faces are shaded yellow by default.

A full partition face can be created when splitting a solid or when using Boolean operations to join multiple solids at shared or intersecting locations.

What You Need To Know Or Remember

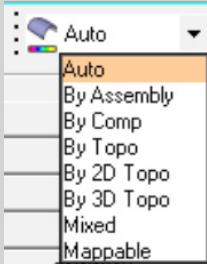
Green edges - 2 surfaces are stitched together; the FE mesh will be linked (compatible), its nodes will line up with the green edge.

Red edges - indicates free surface edges. Red edges inside the geometry tell you that the surfaces are not stitched together (gap); the FE mesh will NOT be linked (not compatible).

Yellow edges – minimum of 3 edges are stitched together; the FE mesh will be compatible.

Blue edges – Suppressed green edge. Surfaces are “melted” together. In other words, the mesher does not see this edge and elements are placed across it.

How to visualize the edge colors?

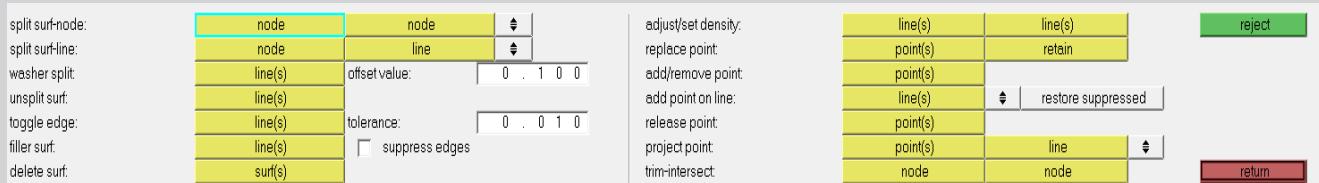


Display is controlled in the Visualization toolbar by activating for instance By 2D Topo (surfaces turn into grey, edges are colored respectively) or Mixed (surfaces are displayed in their original color (reminder: surface color is controlled in the Model Browser), edges are colored respectively).

Panels to be used:

- Toggle surfaces (combining, stitching)
- Trimming surfaces (splitting)
- Suppressing combined edges

Geometry > Quick Edit opens up a very comprehensive panel which allows you (among many other options) to execute the above listed tasks.



The before mentioned panels describe just a very minor fraction of HyperMesh's geometry cleanup functionalities.

Once you feel more comfortable with the process, you will automatically explore and learn more about other techniques.

8.3 **Geometry Creation And Editing**

There are many different ways to create geometry in HyperMesh which include importing geometry from external CAD models, as well as creating new geometry from scratch. The methods used to create a particular geometry depend on both the entities available for input and the level of detail required.

The following is a list of the geometric entities which can be created and edited within HyperMesh:

- Nodes
- Free Points
- Fixed Points
- Lines
- Faces
- Surfaces
- Solids

For each of these entities, we will investigate how they can be created.

Nodes

A node is the most basic finite element entity. A node represents a physical position on the structure being modeled and is used by an element entity to define the location and shape of that element. It is also used as temporary input to create geometric entities.

A node may contain a pointer to other geometric entities and can be associated directly to them. For example, for a node to be translated along a surface, it must first be associated to the surface.

A node is displayed as a small circle or sphere, depending on the mesh graphics mode. Its color is always yellow.

Nodes are created using the menu bar by selecting Geometry > Create > Nodes and then selecting a method to create the nodes with.

Free Points

A free point is a zero-dimensional geometry entity (for more information see: HyperMesh> HyperMesh Entities & Solver Interfaces > Collectors and Collected Entities in the help) in space that is not associated with a surface.

It is displayed as a small “x” and its color is determined by the component collector to which it belongs. These types of points are typically used for weld locations and connectors.

Free points are created using the menu bar by selecting Geometry > Create > Free Points and then selecting a method to create the free points with.

Fixed Points

A fixed point is a zero-dimensional geometry entity in space that is associated with a surface. Its color is determined by the surface to which it is associated.

It is displayed as a small “o”. The automesher places an FE node at each fixed point on the surface being meshed. These types of points are typically used for weld locations and connectors.

Fixed points are created using the menu bar by selecting Geometry > Create > Fixed Points and then selecting a method to create the fixed points with.

Lines

A line represents a curve in space that is not attached to any surface or solid. A line is a one-dimensional geometric entity. The color of a line is determined by the component collector to which it belongs.

A line can be composed of one or more line types. Each line type in a line is referred to as a segment. The end point of each line segment is connected to the first point of the next segment. A joint is the common point between two line segments. Line

segments are maintained as a single line entity, so operations performed on the line affect each segment of the line. In general, HyperMesh automatically uses the appropriate number and type of line segments to represent the geometry.

It is important to realize that lines are different from surface edges and are sometimes handled differently for certain HyperMesh operations.

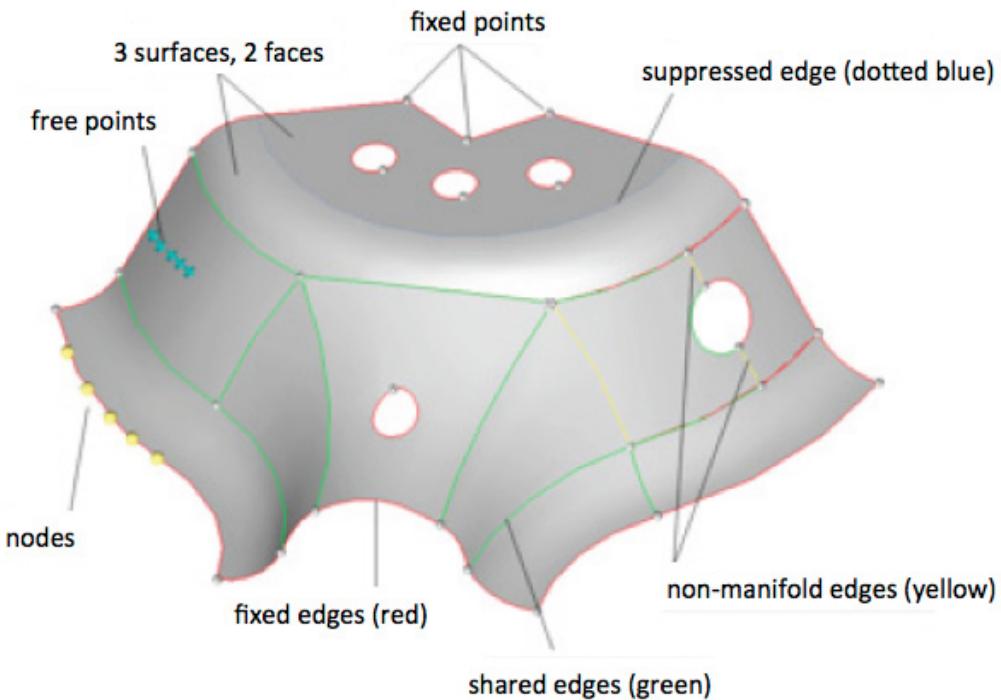
Lines are created using the menu bar by selecting Geometry > Create > Lines and then selecting a method to create the lines with.

Surfaces

A surface represents the geometry associated with a physical part. A surface is a two-dimensional geometric entity that may be used in automatic mesh generation. Its color is determined by the component collector to which it belongs.

A surface is comprised of one or more faces. Each face contains a mathematical surface and edges to trim the surface, if required. When a surface has several faces, HyperMesh maintains all of the faces as a single surface entity. Operations performed on the surface affect all the faces that comprise the surface. In general, HyperMesh automatically uses the appropriate number of and type of surface faces to represent the geometry.

Surfaces are created using the menu bar by selecting Geometry > Create > Surfaces and then selecting a method to create the surfaces with.



Solids

A solid is a closed volume of surfaces that can take any shape. Solids are three-dimensional entities that can be used in automatic tetra and solid meshing. Its color is determined by the component collector to which it belongs.

The surfaces defining a solid can belong to multiple component collectors. The display of a solid and its bounding surfaces are controlled only by the component collector to which the solid belongs.

Solids are created using the menu bar by selecting Geometry > Create > Solids and then selecting a method to create the solids with.

Reminder: The newly created geometry will be placed /stored in the currently active component collector.
Check the Model Browser for the current component collector.

8.4 Importing CAD Geometry

To import geometry, the **Import Browser**, accessible through the Import Geometry icon  , is used.

Using the **Import** Browser, the user can import data from popular CAD packages such as (HyperMesh 13.0)

- ACIS
- CATIA V4/V5
- CATIA Composites Link
- DXF
- FiberSim
- IGES
- JT
- Parasolid
- PDGS
- Pro E
- SolidWorks
- STEP
- Tribon
- UG
- VDAFS

On **Export** HyperMesh 13.0 supports:

- CATIA Composites Link
- FiberSim
- IGES
- JT
- Parasolid
- STEP

Advanced Import Options

The cleanup tolerance is used to determine if two surface edges are the same and if two surface vertices are the same. The cleanup tol toggle controls the following items:

- if two surface edges are close enough to be automatically combined into a shared edge (green edges)
- if a surface is degenerate and should be removed.

If you use the automatic cleanup tolerance option, the complexity of the surface and edge geometries are taken into account and a tolerance to maximize the shared edges (green edges) is selected. The automatic cleanup tolerance value defaults to 100 times what is used internally by the translator.

If you want to specify a different value, use the **manual** cleanup tolerance option, which must be greater than the default value. The translator modifies data only if the data stays within the original data tolerance. Increasing the tolerance can cause serious problems. When this value is set, any features equal to or less than the tolerance are eliminated. The translator does not include any edge less than the tolerance long; if there are edges present that are important to the surface, that surface will be distorted, or will fail to trim properly. Surfaces smaller than the tolerance may not be imported. If the file you have read has many very short edges, it may be worthwhile to reread the file using a larger tolerance. The same holds true if surfaces appear to be “inside out” when surface lines are displayed. The tolerance value should not be set to a value greater than the node tolerance (set in the Options panel) to be used for your element mesh.

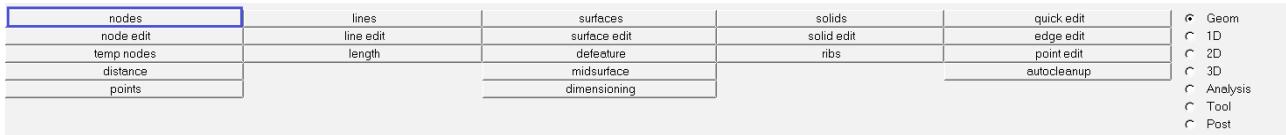
The Options panel is accessed through the menu bar by selecting Preferences > Geometry Options.

- The Import blanked (no show) components option allows you to control if blanked components in the IGES translator will be imported into HyperMesh, as well as components containing “NO SHOW” entities from the Catia translator.
- Place a check in the Name components by layer option to activate this option. This option is valid for Catia V4 and Catia V5.
- For Catia V4, the option is enabled by default, and can't be disabled.
- For Catia V5, the option is disabled by default and can be enabled. If this option is enabled, Catia objects from the same layer are grouped into the same component.

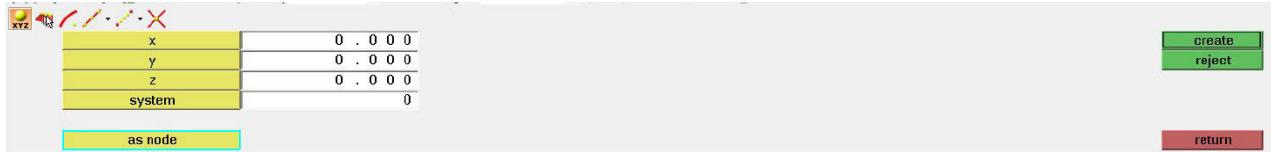
8.5 Geometry FAQ's

1. How to find coordinates of a node? How to get X,Y,Z coordinates of a node?

Go to Geom > nodes



Click on “As node” option and then click on the node

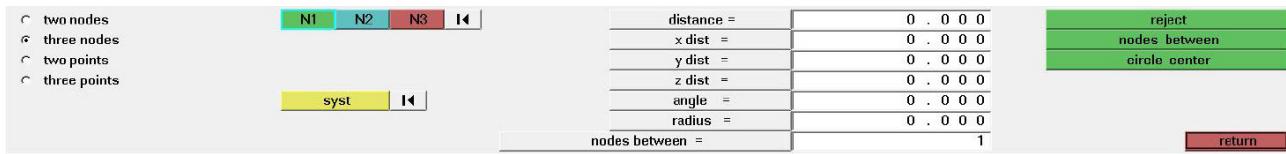


2. How to create a node at the centre of a circle? How to measure the radius of the hole?

Go to Geom-->Distance



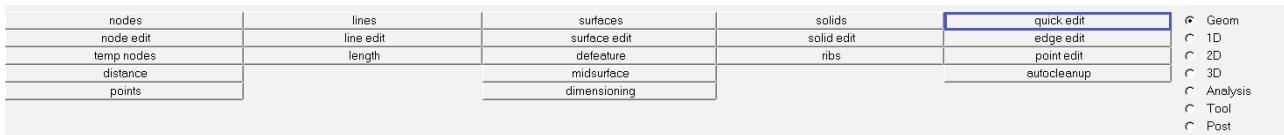
Go to Three nodes->



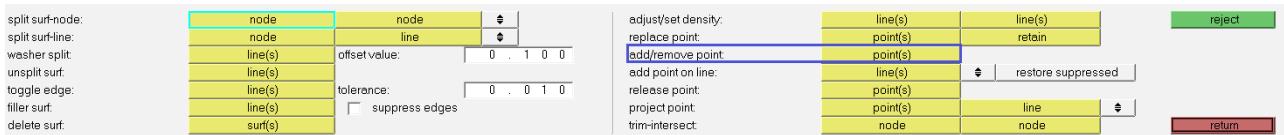
Select three nodes on the circumference of the circle > circle centre

3. How to remove fixed points from the surface?

Go to Geom > quick edit panel

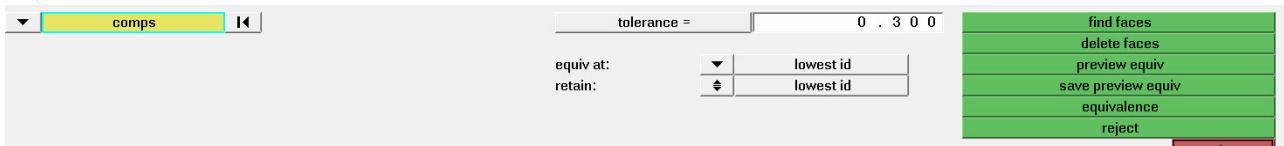


add/remove points option

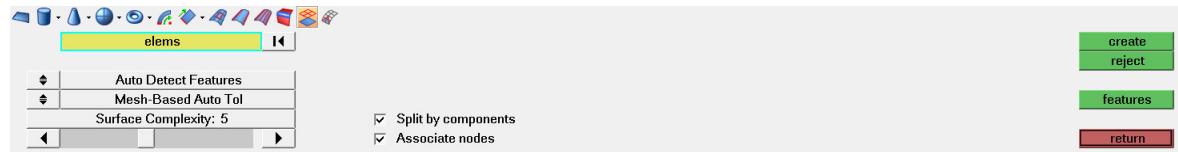


4. How to export the geometry representing the FE model as iges/create surfaces from FE elements?

First extract faces using tool>faces



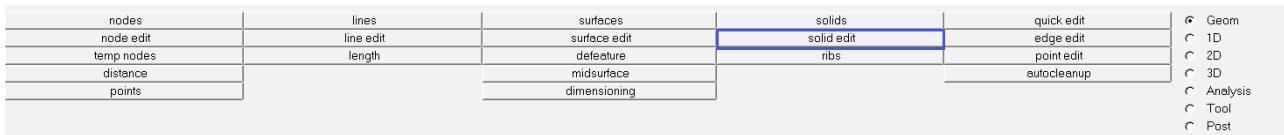
then go to geometry>surfaces>from FE



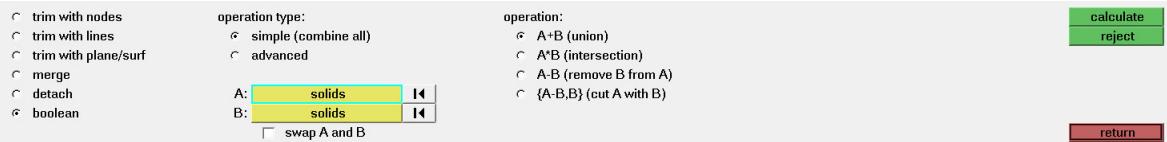
User must either create feature lines or specify appropriate surface complexity while asking HM to recognize feature lines by itself, feature lines help HM decide where to end one surface and start another.

5. How to merge solids/subtract one solid space from another?

Geom > Solid edit > Boolean



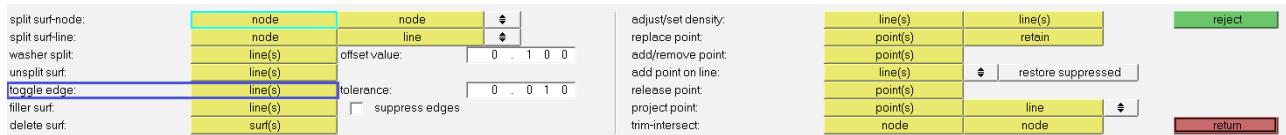
Use this subpanel to perform complex trimming and merging operations, such as trimming a pair of intersecting solids down to only the portions of them that coincide.



Use (A+B) union. This involves combining multiple solids into single solids.

6. How to change the edge type from free to shared or shared to suppressed in a surface?

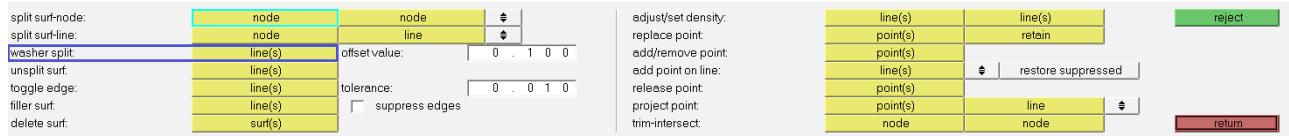
Go to Geom > quick edit > toggle edge



Left-click a free edge (red) to make it shared (green), or a shared edge to make it suppressed (blue). Right-click a suppressed edge to make it shared, or a shared edge to make it free.

7. How to create washers on geometry and enter its dimensions ?

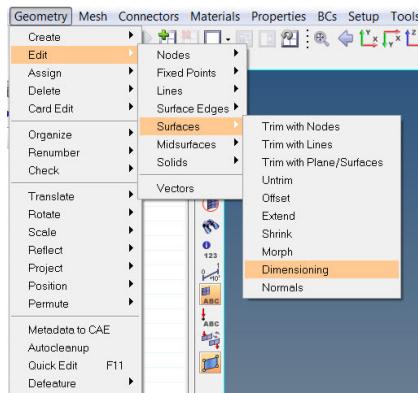
Go to Geom > quick edit > washer split Use these controls to create a washer seated on a hole in your model.



First specify an offset to determine the width of the washer, and then highlight the line selector and pick the line(s) to which you wish to add washers of that thickness

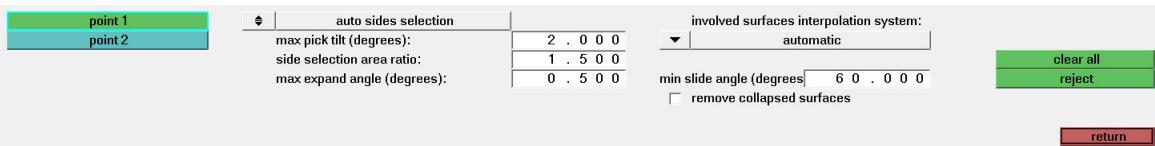
8. How to change one or more dimensions of existing geometry

Click Geometry > Edit > Surfaces > Dimensioning to open the Dimensioning panel.



Click the point in the model for point1.

Click the point in the model for point2.



A dimension manipulator with a value is created between the points, which represents the thickness of the part.
Click the dimension text for the dimension manipulator created in the previous step.

Enter the new value and press enter.

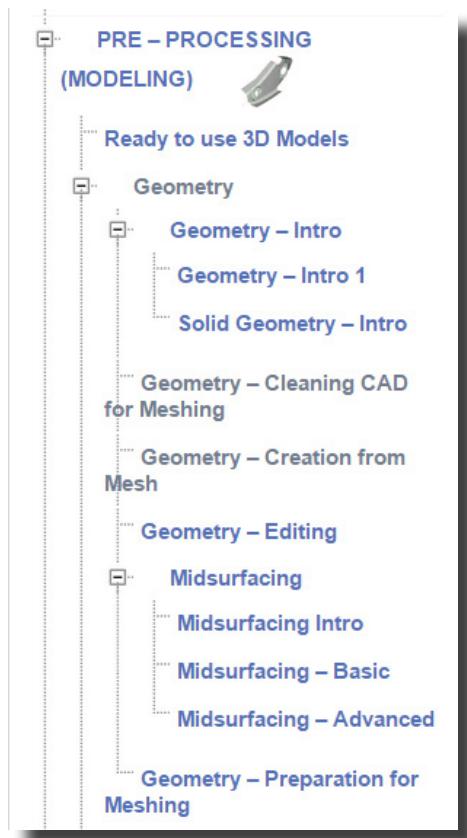
8.6 Recommended Tutorials And Videos

Recommended Tutorials

These selected tutorials are part of the HyperWorks installation (see Help -> Tutorials ...):

- HM-2000: Importing and Repairing CAD
- HM-2010: Generating a Midsurface
- HM-2020: Simplifying Geometry
- HM-2030: Refining Topology to Achieve a Quality Mesh
- HM-2040: Creating and Editing Line Data
- HM-2060: Creating and Editing Solid Geometry
- HM-2070: Geometry and Mesh Editing Using the Quick Edit Panel
- HM-2080: Modifying Models using solidThinking
- HM-2090: Dimensioning

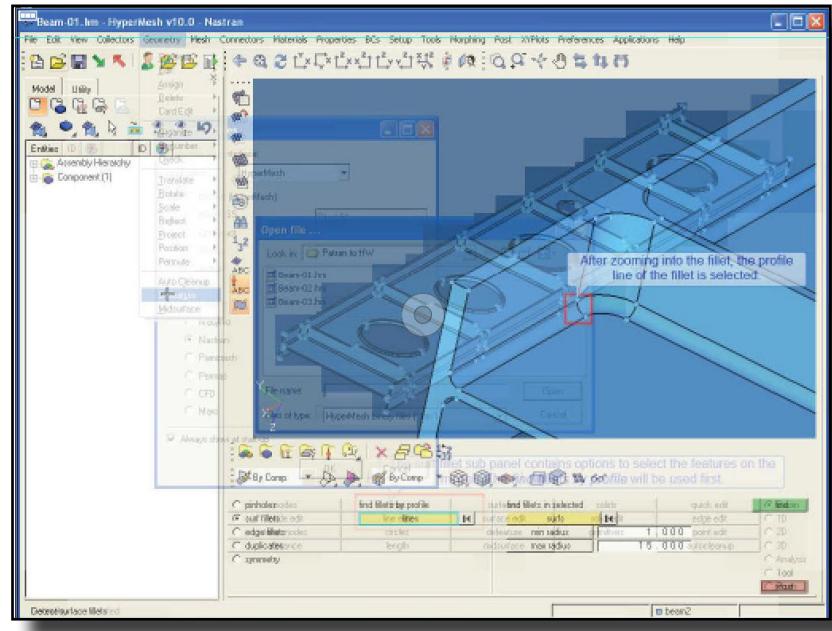
In the **Academic Training Center** (www.training.altairuniversity.com) additional tutorials are available related to creating and editing geometry in HyperMesh



Recommended Videos

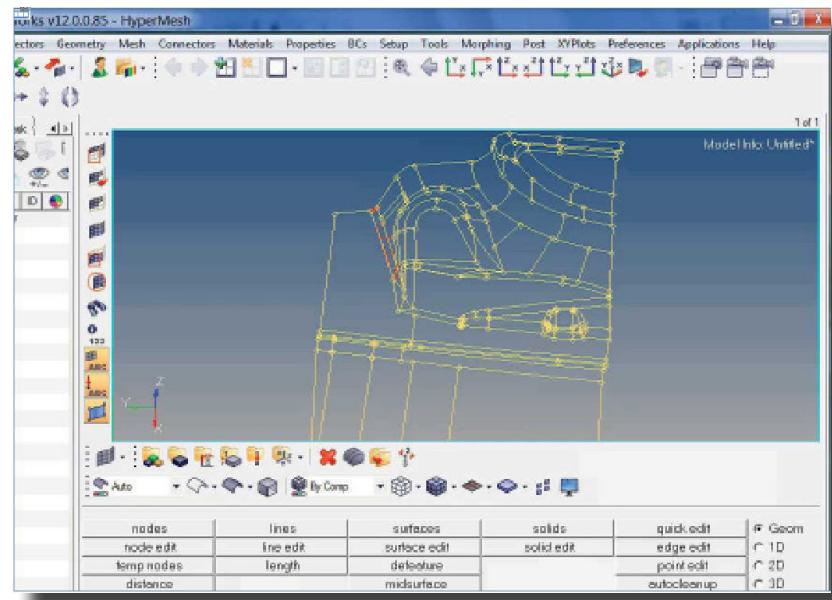
- Editing geometry in HyperMesh**

(<http://altair-2.wistia.com/medias/y0g86ylix2>)

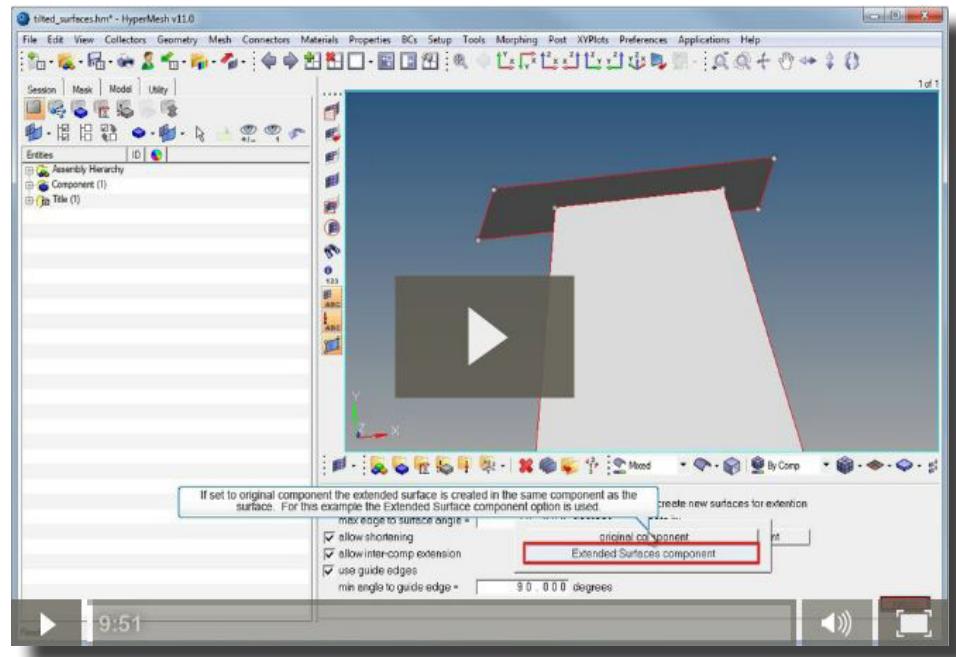


- Geometry cleanup in HyperMesh**

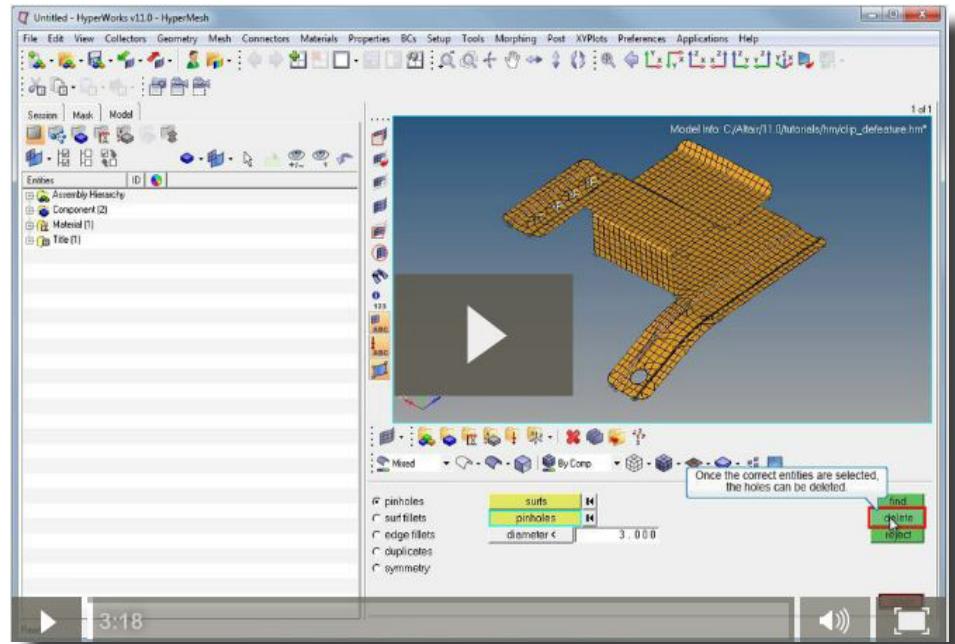
(<http://altair-2.wistia.com/medias/g6mozquujp>)



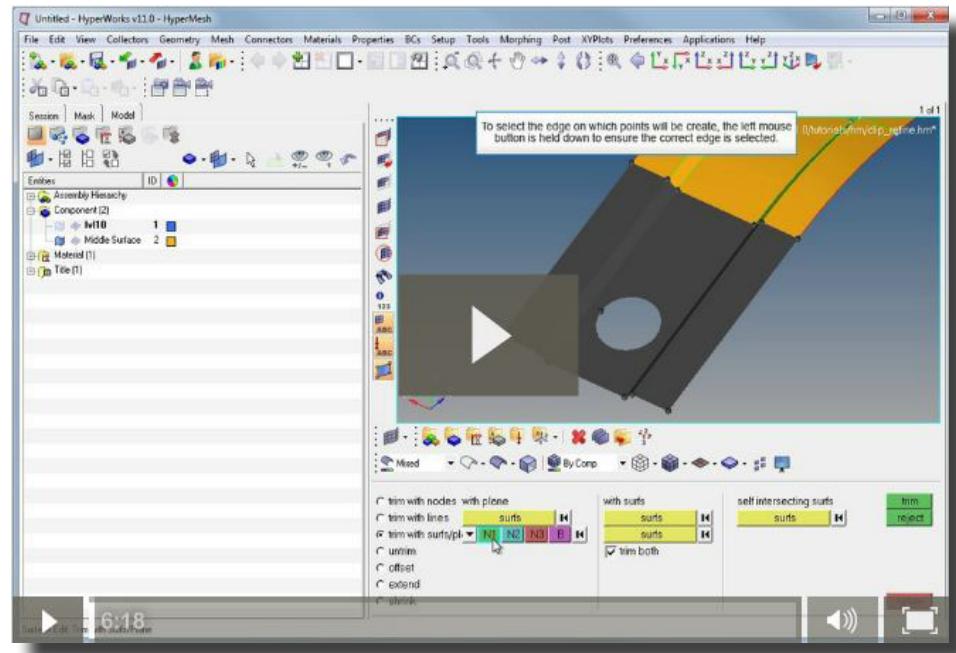
- **Extend surfaces** (<http://altair-2.wistia.com/medias/vu8femddfb>)



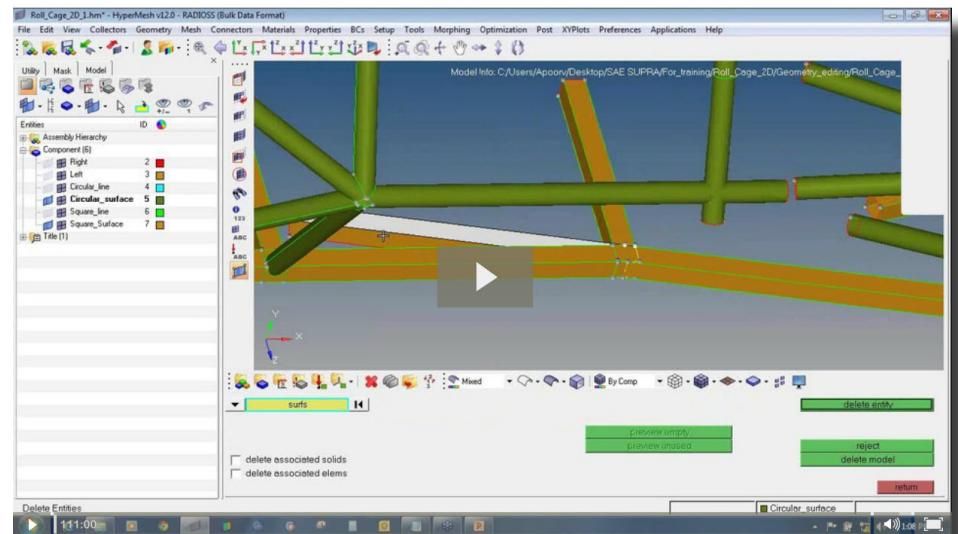
- **Simplify geometry** (<http://altair-2.wistia.com/medias/2ecpq70e15>)



- **Refining topology** (<http://altair-2.wistia.com/medias/2p48j67uav>)



- **Surface creation and geometry editing of Roll Cage** (and 2D Meshing, by Apoorv Bapat)



<https://altair-2.wistia.com/medias/35ncsicmuo>

You may also consider to attend a HyperWorks class held in one of our facilities. For students and teachers the seminar fee is typically highly discounted - sometimes even waived.

Just contact your local Altair University Liaison for details (altairuniversity@altair.com).

8.7 Student Racing Car Project - Introduction And CAD Related Aspects

Student Race Car Introduction

The objective of this demo is to provide insight into the typical modeling steps related to the analysis of a student racing car subjected to bending-, shear-, and torsion loads.

Please note, that the load cases, geometry of the frame, the dimensions of the 1D element cross sections, material parameters etc. are just **conceptual**.



The following working steps will be briefly discussed in the course of this book. The focus of this chapter is on CAD.

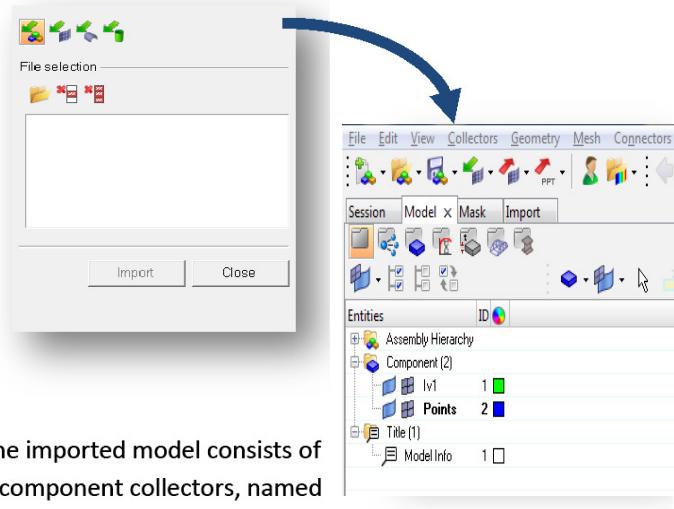
- [Model geometry \(CAD\) import](#)
- 1D meshing (CBAR elements)
- 1D mesh checks
- Material definition (Card Image MAT1)
- Definition of frame properties (Card Image PBARL) & trouble shooting
- Wheel suspension
- Boundary conditions & loads
- Analysis
- Postprocessing

CAD

In order to keep this demo simple, the given CAD was updated and improved in order to account for misaligned lines, duplicated lines, and minor asymmetries in the overall geometry (a more faulty geometry is available too).

Let's get started with importing the CAD data.

The imported model consists of 2 component collectors that are named `lv1` and `Points` (compare with the component collectors listed in the Model Browser).



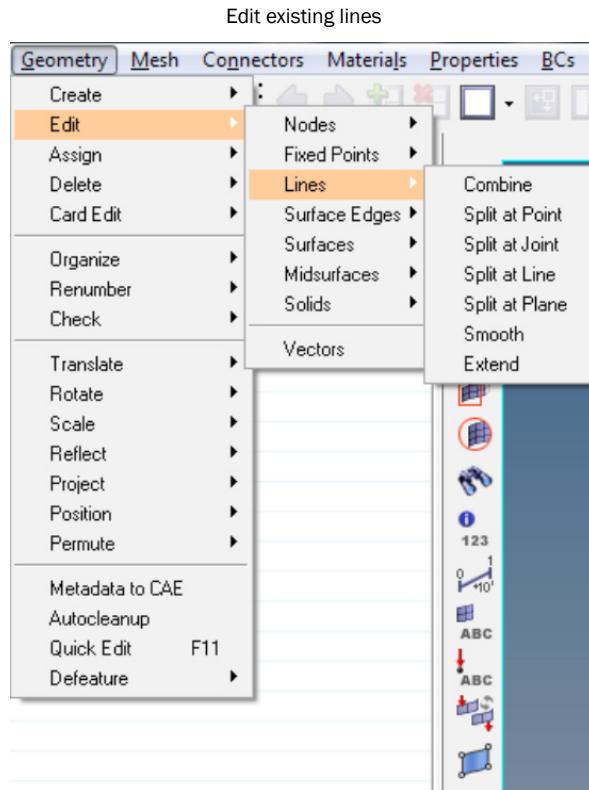
The imported model consists of 2 component collectors, named "lv1" and "Points".

The image below indicates that the geometry of the student race car frame consists of lines only. The crosses displayed at the location of the lines indicate what are called "line handles". These ease the viewing and selection of lines.

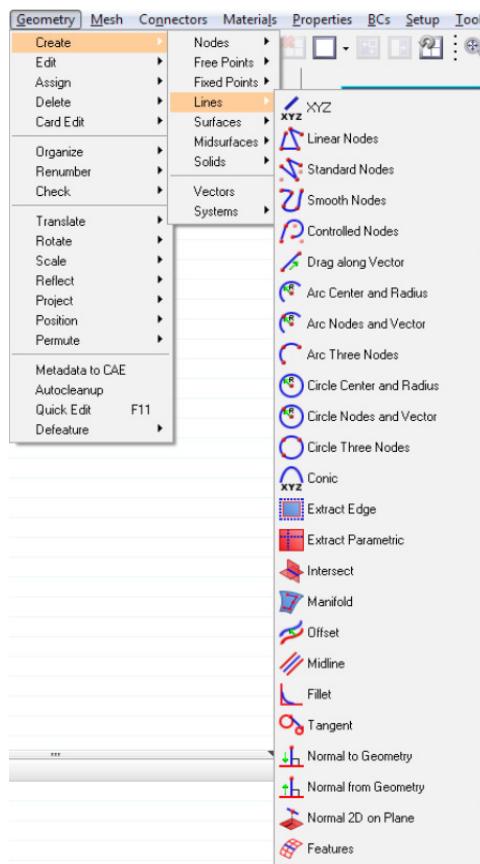


In the image the "location" of the wheel suspension is given in terms of pre-defined geometry points (x symbols).

Depending on the quality of the given CAD data (here: lines) it may be required to manually trim (split), combine or extend lines (Geometry > Edit > Lines > ...)



In case lines need to be created from scratch, there are many possibilities available:

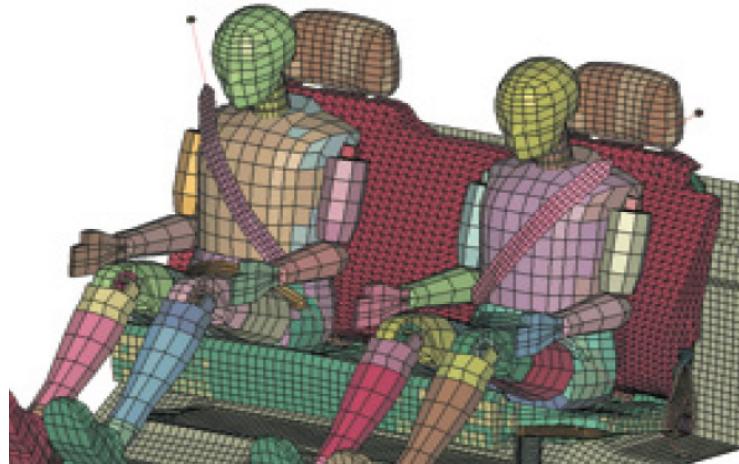


Create Lines From Scratch

9

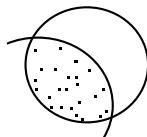
Introduction To Meshing

This chapter includes material from the book “**Practical Finite Element Analysis**”. It also has been reviewed and has additional material added by Gareth Lee.



Courtesy: Tata Johnson Controls Automotive Limited, India

9.1 Why Do We Carry Out Meshing?



No. of Points = ∞
DOF per Point = 6
Total equations = ∞



No. of Nodes = 8
DOF per Node = 6
Total equations = 48

The basic idea of FEA is to make calculations at only limited (Finite) number of points and then interpolate the results for the entire domain (surface or volume). Any continuous object has infinite degrees of freedom and it's just not possible to solve the problem in this format. Finite Element Method reduces the degrees of freedom from infinite to finite with the help of discretization or meshing (nodes and elements).

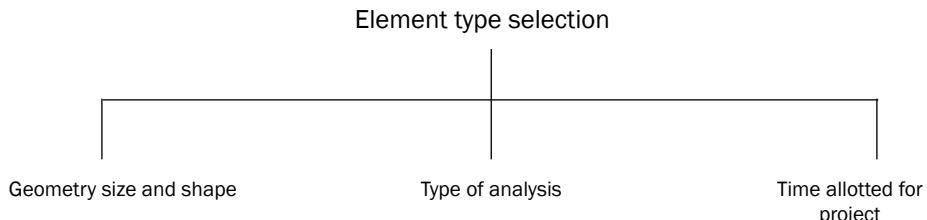
9.2

Types Of Elements

Elements			
1D	2D	3D	Other
 $x \ggg y, z$	 $x, z \ggg y$	 $x \sim y \sim z$	
<p>One of the dimensions is very large in comparison to the other two</p> <p><u>Element shape</u> – line</p> <p><u>Additional data from user</u> - remaining two dimensions i.e. area of cross section</p> <p><u>Element type</u> – rod, bar, beam, pipe, axisymmetric shell, etc</p> <p><u>Practical applications</u> - Long shafts, beams, pin joint, connection elements, etc.</p>	<p>Two of the dimensions are very large in comparison to the third one</p> <p><u>Element shape</u> – quad, tria</p> <p><u>Additional data from user</u> - remaining dimension i.e. thickness</p> <p><u>Element type</u> – thin shell, plate, membrane, plane stress, plane strain, , axisymmetric solid, etc.</p> <p><u>Practical applications</u> - Sheet metal parts, plastic components like an instrument panel ,etc.</p>	<p>All dimensions are comparable</p> <p><u>Element shape</u> – tetra, penta, hex, pyramid</p> <p><u>Additional data from user</u> - none</p> <p><u>Element type</u> – solid</p> <p><u>Practical applications</u> - Transmission casing, engine block, crankshaft, etc.</p>	<p><u>Mass</u> – Point element, concentrated mass at the center of gravity of the component</p> <p><u>Spring</u> – translational and rotational stiffness</p> <p><u>Damper</u> - damping coefficient</p> <p><u>Gap</u> – Gap distance, stiffness, friction</p> <p><u>Rigid</u> – RBE2, RBE3</p> <p><u>Weld</u></p>

9.3

How To Decide The Element Type

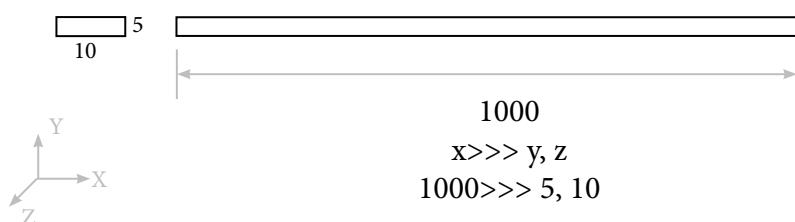


A. Geometry Size And Shape

For an analysis, the software needs all three dimensions defined. It can not make calculations unless the geometry is defined completely (by meshing using nodes and elements).

The geometry can be categorized as 1D, 2D, or 3D based on the dominant dimensions and then the type of element is selected accordingly.

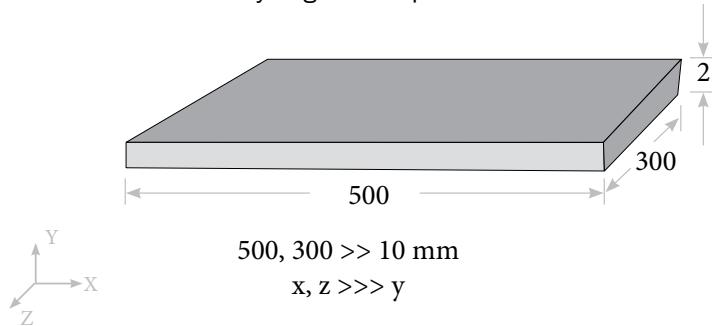
1D Element: Used for geometries having one of the dimensions that is very large in comparison to the other two.



The shape of the 1D element is a line. When the element is created by connecting two nodes, the software knows about only one out of the 3 dimensions. The remaining two dimensions, the area of the cross section, must be defined by the user as additional input data and assigned to the respective elements

Practical example: Long shaft, rod, beam, column, spot welding, bolted joints, pin joints, bearing modeling, etc.

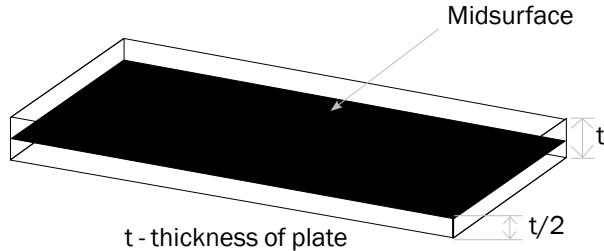
2D Element: Used when two of the dimensions are very large in comparison to the third one.



2D meshing is carried out on a mid surface of the part. 2D elements are planar, just like paper. By creating 2D elements, the software knows 2 out of the 3 required dimensions. The third dimension, thickness, has to be provided by the user as an additional input data.

Why Is 2D Meshing Carried Out On A Mid Surface?

Mathematically, the element thickness specified by the user is assigned half on the element top and half on the bottom side. Hence, in order to represent the geometry appropriately, it is necessary to extract the mid surface and then mesh on the mid surface.



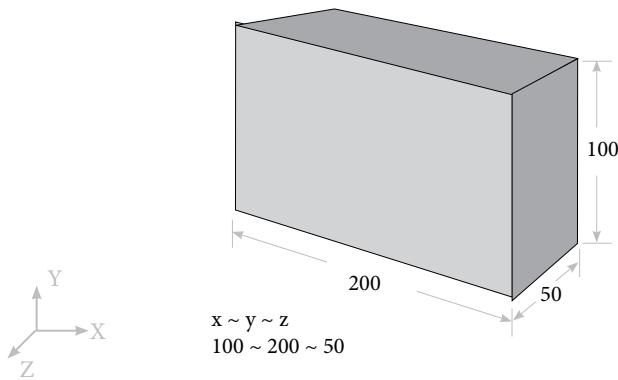
Practical example: All sheet metal parts, plastic components like instrument panels, etc. In general, 2D meshing is used for parts having a width / thickness ratio > 20 .

Limitations Of Mid Surface And 2D Meshing

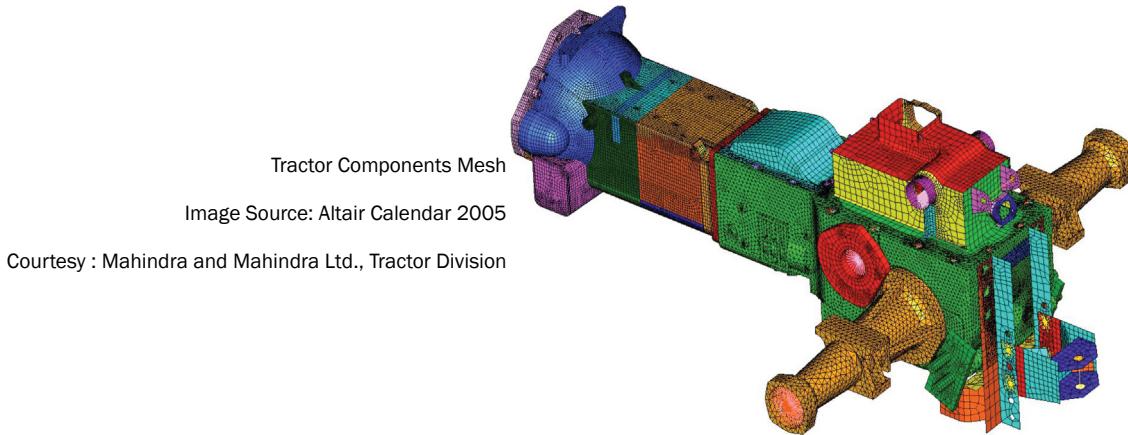
2D meshing would lead to a higher approximation if used for

- variable part thickness
- surfaces are not planar and have different features on two sides.

3D Element: used when all three dimensions are comparable



Practical examples: Transmission casing, clutch housing, engine block, connecting rod, crank shaft etc.



B. Based On The Type Of Analysis

Structural and fatigue analysis - Quad, hex elements are preferred over trias, tetras and pentas.

Crash and nonlinear analysis – Priority to mesh flow lines and brick elements over tetrahedron.

Mold flow analysis – Triangular element are preferred over quadrilateral.

Dynamic analysis – When the geometry is borderline between the classification of 2D and 3D geometry, 2D shell elements are preferred over 3D. This is because shell elements being less stiff captures the mode shapes accurately and with a fewer number of nodes and elements.

C. Time Allotted For Project

When time is not a constraint, the appropriate selection of elements, mesh flow lines, and a good mesh quality is recommended. Sometimes due to a very tight deadline, the analyst is forced to submit the report quickly. For such situations

1. Automatic or batch meshing tools could be used instead of time consuming but structured and good quality providing methods.
2. For 3D meshing tetras are preferred over hexas.
3. If the assembly of several components is involved then only the critical parts are meshed appropriately. Other parts are either coarse meshed or represented approximately by 1D beams, springs, concentrated mass, etc.

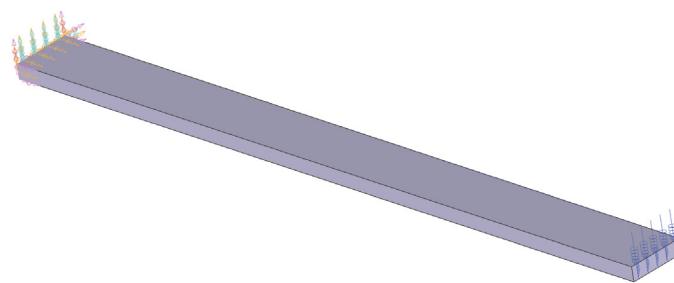
9.4

Can We Solve The Same Problem Using 1D, 2D And 3D Elements

Is it not possible to use 3D elements for long slender beams (1D geometry), for sheet metal parts (2D geometry), and 2D shell elements for representing big casting parts ?

The same geometry could be modelled using 1D, 2D, or 3D elements. What matters is the number of elements and nodes (DOF), the accuracy of the results, and the time consumed in the analysis.

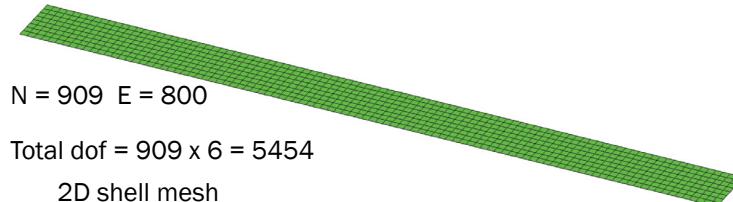
For example, consider a cantilever beam with a dimension of 250 x 20 x 5 mm that is subjected to a 35 N force:



1D beam model

$N=2 \ E=1$

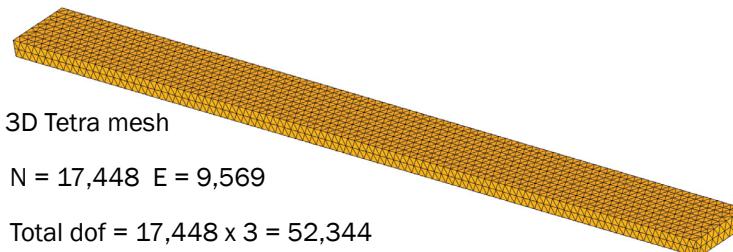
Total DOF = $6 \times 2 = 12$



$N = 909 \ E = 800$

Total dof = $909 \times 6 = 5454$

2D shell mesh



3D Tetra mesh

$N = 17,448 \ E = 9,569$

Total dof = $17,448 \times 3 = 52,344$

	Nodes	Elements	Stress N/mm ²	Displacement mm
Analytical	--	--	105	4.23
1D	2	1	105	4.23
2D	909	800	103	4.21
3D	17,448	9,569	104	4.21

9.5 How To Decide Element Length

- Based on previous experience with a similar type of problem (successful correlation with experimental results).
- Type of analysis: Linear static analysis could be easily carried out quickly with a large number of nodes and elements, but crash, nonlinear, CFD, or dynamic analysis takes a lot of time. Keeping control on the number of nodes and elements is necessary.
- Hardware configuration and graphics card capacity of the available computer. An experienced CAE Engineer knows the limit of the nodes that can be satisfactorily handled with the given hardware configuration.

Suppose you are a part of a newly formed CAE group (no clear guidelines are available, and there is no experienced person in the group): In the first run, accept the default element length. Mesh with the basic rules of thumb discussed in this book. Then run the analysis and observe the high stress regions. Remesh the localized areas of high stress (with smaller element length) and solve again. Compare the difference in the original and the new results. Continue the process until convergence is achieved (5 to 10% difference in strain energy / maximum stress value).

9.6 How To Start Meshing

1) Spend A Sufficient Amount Of Time Studying The Geometry

A common observation is that CAE engineers start meshing immediately, without properly understanding the geometry and paying attention to all of the requirements and instructions provided. Observing the geometry several times and thinking about it from all angles is strongly suggested. Mental visualization of the steps is the first step in the right direction of creating a good meshing.

2) Time Estimation

A time estimation is very relative and one can find a lot of differences in estimation by different engineers (as much as 2 to 3 times). Usually a less experienced person will estimate more time. Also if someone is handling the job for the first time, then he/she will require more time. If similar kinds of jobs are given to the same engineer again and again, the meshing time would reduce drastically.

3) Geometry Check

Generally CAD data is provided in *.igs format. Geometry cleanup is an integral part of the meshing activity. CAE engineers should at least have the basic knowledge of CAD. Before starting the job, the geometry should be carefully checked for:

- Free edges
- Scar lines
- Duplicate surfaces
- Small fillets
- Small holes
- Beads
- Intersection of parts (assembly of components)

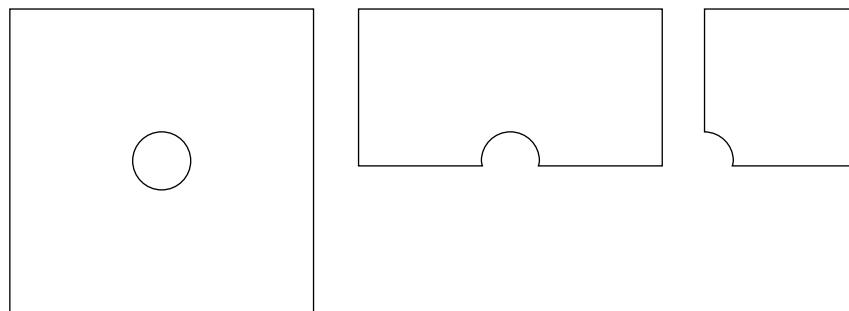
If suppressing fillets, small holes, beads, or the generation of a mid surface is required for meshing, then why isn't the CAD data provided in the way needed for CAE by the CAD engineers?

Yes, theoretically that would be an ideal situation, but practically everyone works with a very tight schedule and target dates. CAD data is generated keeping in mind the final drawing to be released for manufacturing. The same CAD model is provided simultaneously to the tools and jig /fixture manufactures, vendors, purchase engineers, and CAE engineers, etc.

The simplification required for a FEA is understood better by a CAE engineer than a CAD engineer. All meshing software provides special tools for geometry cleanup and simplification, which are usually much faster than CAD software. Many times, for complicated geometry, surfacing operations fail in CAD software and it could be easily handled by the CAE engineer by avoiding the geometry and generating the mesh using manual or special meshing operations.

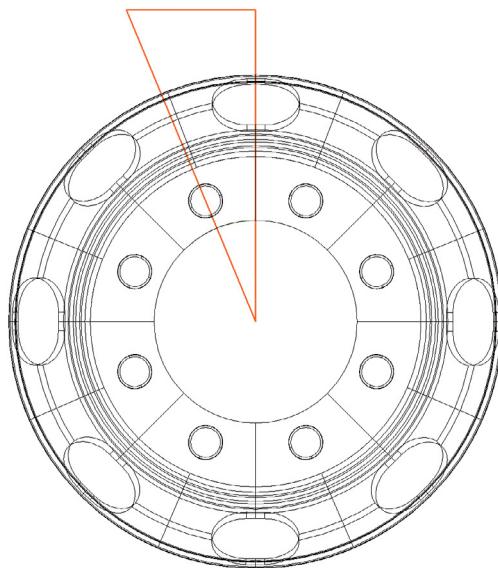
4) Symmetry Check

- Complete part symmetry



Meshing only a quarter of the plate and reflecting it twice is advisable.

- Sub-part symmetry, repetition of features, and the copy/paste command

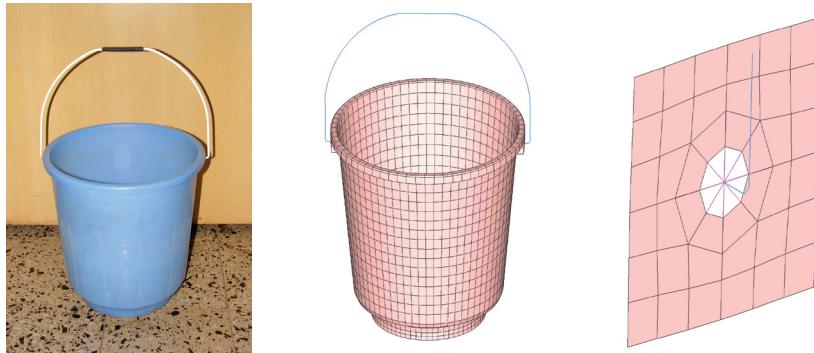


Meshing the highlighted 22.5° portion and then using reflection and rotation would lead to a faster mesh as well as the same structure of elements and nodes around the critical areas (holes).

5) Selection Of Type Of Elements

In real life, we rarely use only one type of element. It is usually a combination of different types of elements (1D, 2D, 3D, and others).

In the above figure, the handle of the bucket is modelled by beam (1D) elements, the bucket body uses shell (2D) elements, and



the connection between the handle and the bucket body through RBE2 (rigid) elements.

6) Type Of Meshing

- Geometry based – The mesh is associated to the geometry. If the geometry is modified, the mesh will also get updated accordingly (automatically). The boundary conditions could be applied on the geometry like a surface or edge, etc.
- FE based – The mesh is non associative. The boundary conditions are applied on the elements and nodes only.

7) Joint Modeling

- a. Special instructions for bolted joints (specific construction around holes)
- b. Spot and arc weld
- c. Contact or gap elements and the requirement of the same pattern on 2 surfaces in the contact
- d. Adhesive joint

8) Splitting The Job

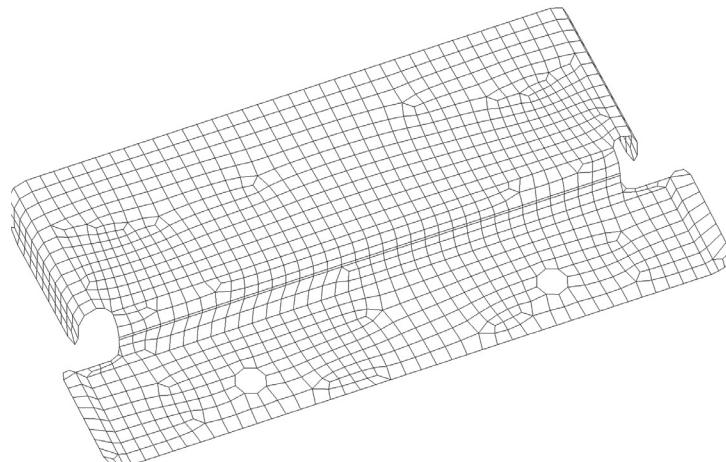
When there is little time or when engineers in other group are sitting idle, then the job could be split among several engineers by providing a common mesh on the interfaces.

9.7 Meshing Techniques

	Automatic / Batch	Mapped (or Interactive)	Manual (Special commands: Spline, Ruled, Drag / extrude, Spin / rotate etc.)
Time required for meshing	↓	≈ (intermediate i.e. more than auto but less than manual)	↑
Geometry required	✓	✓	X
No. of nodes and elements generated	↑	≈	↓
User friendliness	↑	≈	↓
User's control over the mesh	↓	≈	↑
Structural mesh (flow lines)	↓	≈	↑
Experience or skill required	↓	≈	↑
Patience	↓	≈	↑ (specially for brick / hex)

Batch meshing / Mesh adviser – Now a days, all software provide special programs for automatic geometry clean up and meshing with little or no interaction from the user. The user has to specify all the parameters like minimum hole diameter, minimum fillet radius, average and minimum element length, quality parameters, etc. and the software will run a program to produce the best possible mesh by fulfilling all or most of the specified instructions. Though these programs are still in the initial stage and for many applications the output is not acceptable, the research is in progress and its performance will surely improve in the coming years.

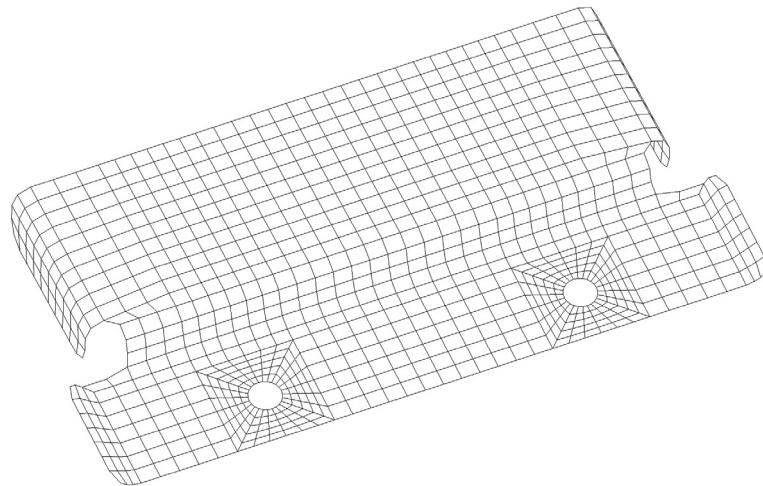
Automatic mesh vs. mapped / interactive / manual mesh



Nodes = 1400, Elements = 1309

Automatic meshing

Not acceptable



Nodes = 1073, Elements = 982

Mapped meshing

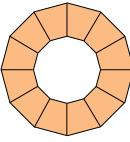
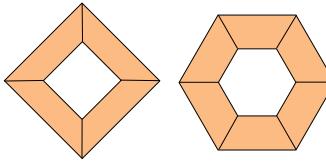
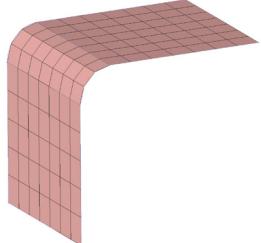
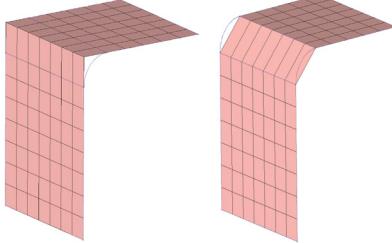
9.8 Meshing In Critical Areas

Critical areas are locations where high stress will occur. Dense meshing and structured mesh (no trias / pentas) is recommended in these regions. Areas away from the critical area are general areas. Geometry simplification and coarse mesh in general areas are recommended (to reduce the total DOFs and solution time).

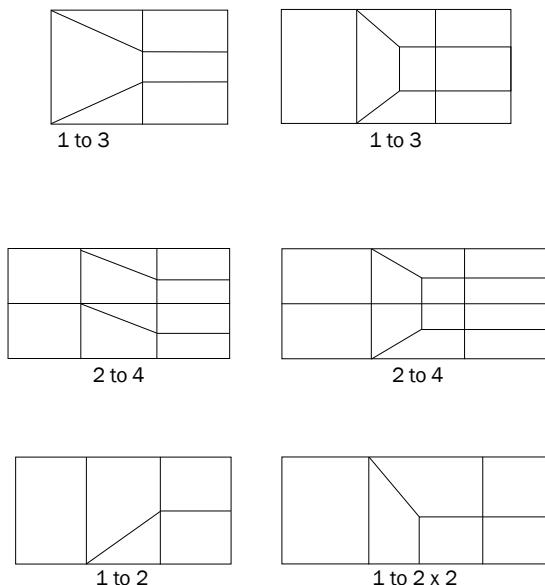
How Would I Know About The Critical Areas Before Carrying Out An Analysis?

After going through a previous analysis of a similar part (carried out by your colleague or a senior in the group) one can get a fairly good idea about the probable locations of the high stress. But suppose there is no past record and you are doing it for the first time, then run the analysis with a reasonable element length and observe the results. High stress regions are critical and could be remeshed with a smaller element length in the second run.

Rules for modeling holes and fillets

Critical area	General area
	 4 to 6 elements
 Minimum 3 elements on fillet.	 Suppress small fillets, 1 element for large fillets.

Mesh transition techniques and flow lines

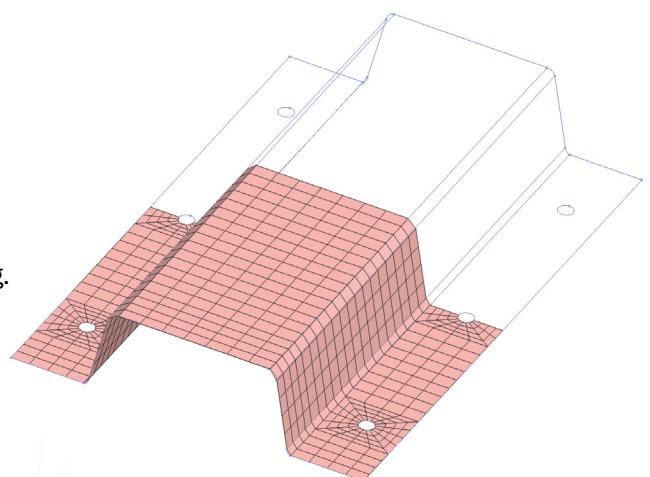


9.9 Mesh Display Options

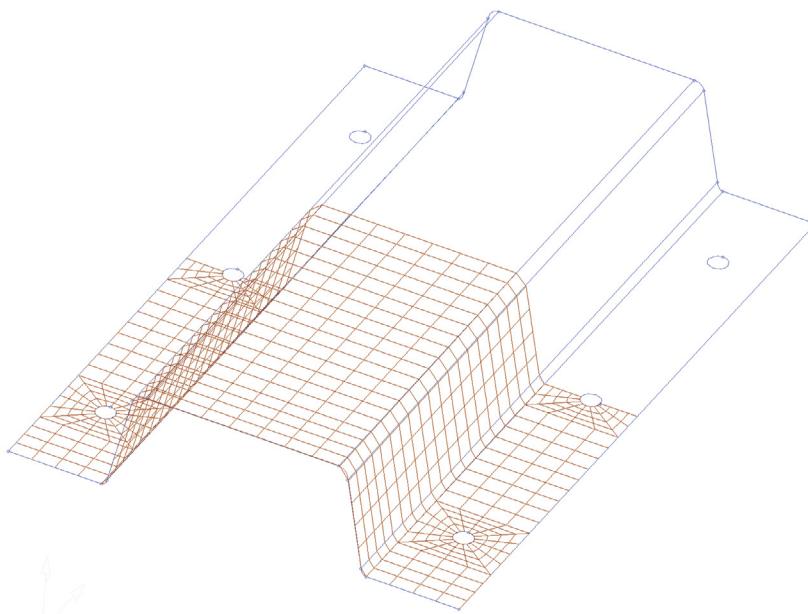
1. Shell Mesh

- a. Mixed mode: Geometry – wireframe, Mesh - shaded

This is the most common and preferred way of working.

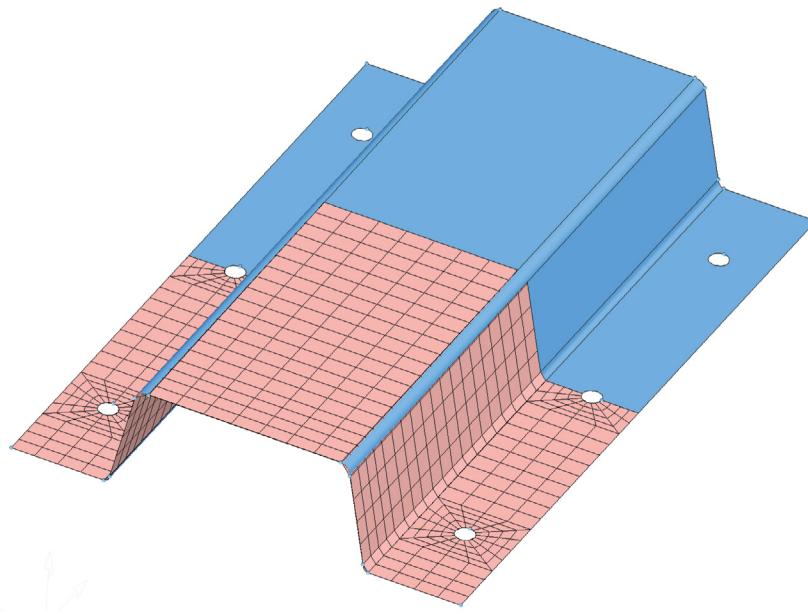


b. Line mode: Geometry and Mesh – both wireframe



This mode is preferred for brick meshing, for internal mesh adjustment / modifications.

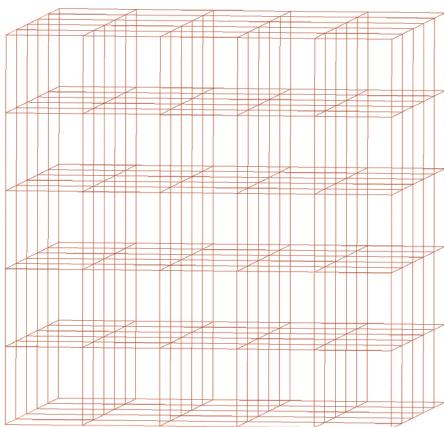
c. Solid: Geometry and Mesh – both shaded



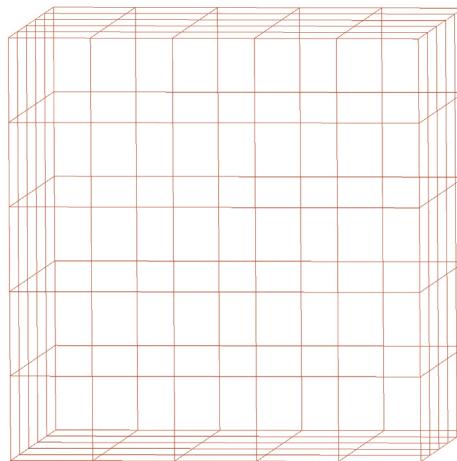
This mode is not preferred for regular meshing but is very useful after the completion of the job. It helps to check the mesh deviation from the geometry and to find the kinks or abrupt changes in the mesh.

2. Brick Mesh

a. Line mode options



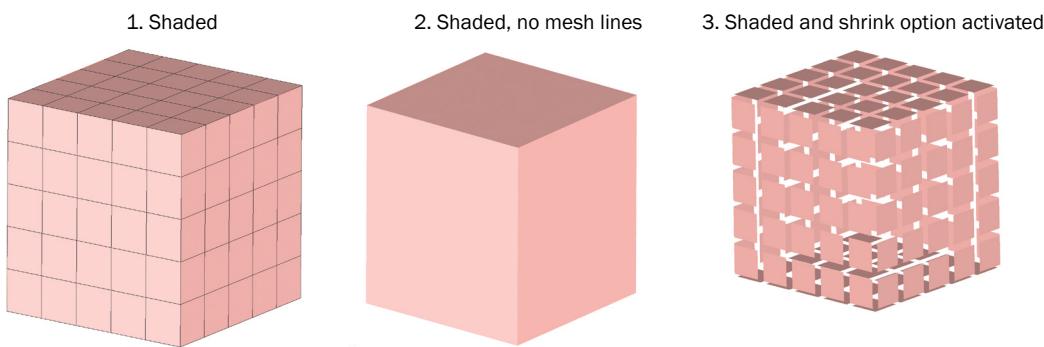
All faces display



Free faces display

The figure on the left is used for viewing the internal details while the figure on the right is used for checking free faces inside the mesh.

b. Shading mode



The solid view is commonly used during regular meshing. The shaded view is used for checking the kinks or deviation of the mesh from geometry and the shrink view is used for checking for free faces and for missing or extra 1D elements on the edge of the element.

9.10 Understanding Element Behavior

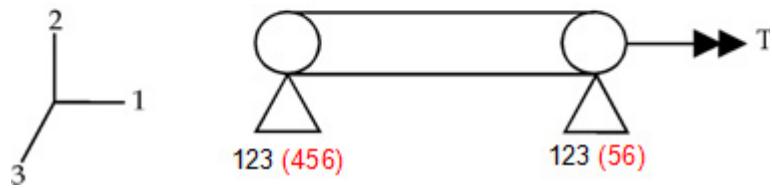
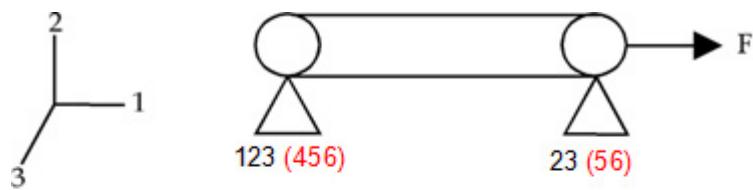
To successfully complete a finite element analysis, you must understand the behavior of various types of elements. A deep theoretical knowledge of element formulations is not necessarily required although a fundamental knowledge of how each element type behaves is essential in the selection of the appropriate element type(s) which will lead to proper interaction with applied loads and boundary conditions.

Finite element models consisting of a single element are one method of studying the mechanics of elements. The inputs and outputs can be studied in detail and compared to solid mechanics solutions. This method is useful for understanding the sign and naming conventions used by a particular solver.

The following diagrams depict single element models, each with several load cases applied in conjunction with a minimum set of boundary conditions. Adding more than the required boundary conditions can be used to learn even more about element behavior.

DOFs are important because they dictate the ability of the elements to model a given problem and also dictate whether or not elements are compatible with each other. Further discussion on element compatibility will follow.

Rod Element

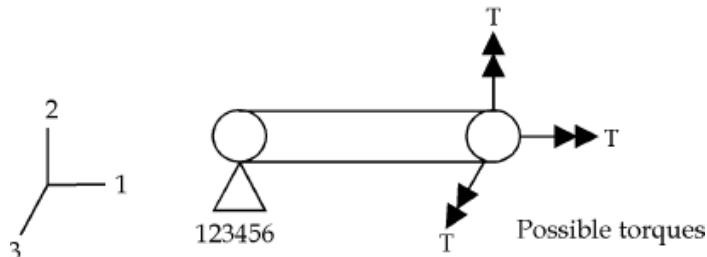
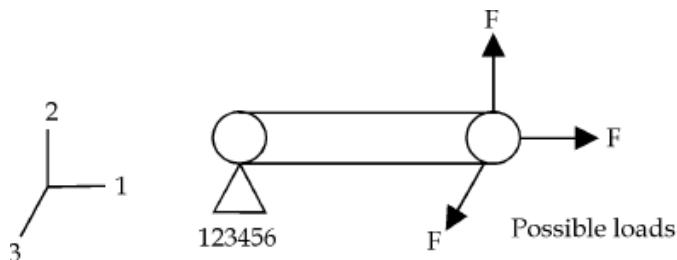


Example of **rod** element

Nodes 2 nodes

DOFs 3 or 6 degrees of freedom per node

Beam Element

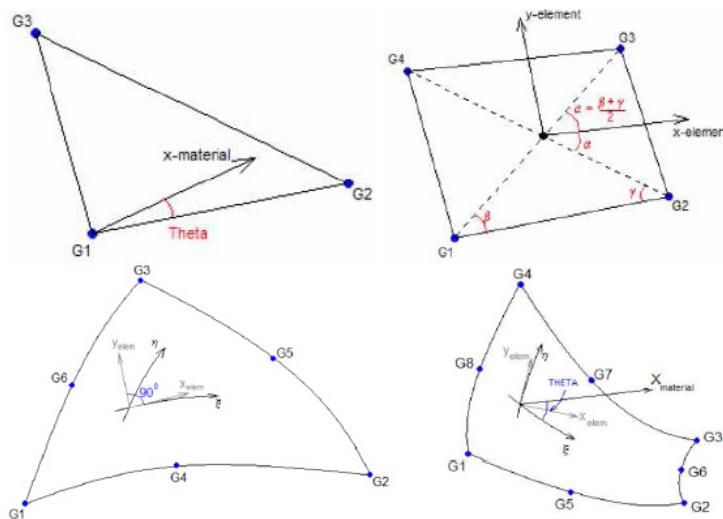


Example of **beam** element

Nodes 2 nodes

DOFs 6 degrees of freedom per node

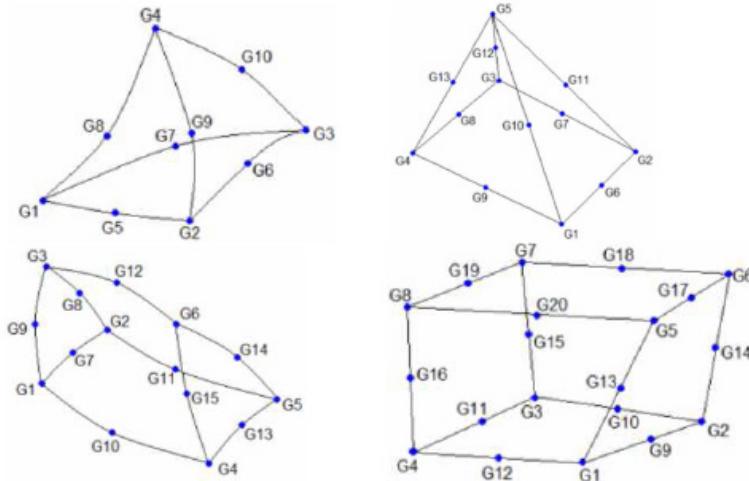
Shell Element



Example of **shell** elements (CTRIA3, CQUAD4, CTRIA6, CQUAD8)

First Order	4 or 3	nodes
Second Order	6 or 8	nodes
DOFs	6	degrees of freedom per node

Solid Element



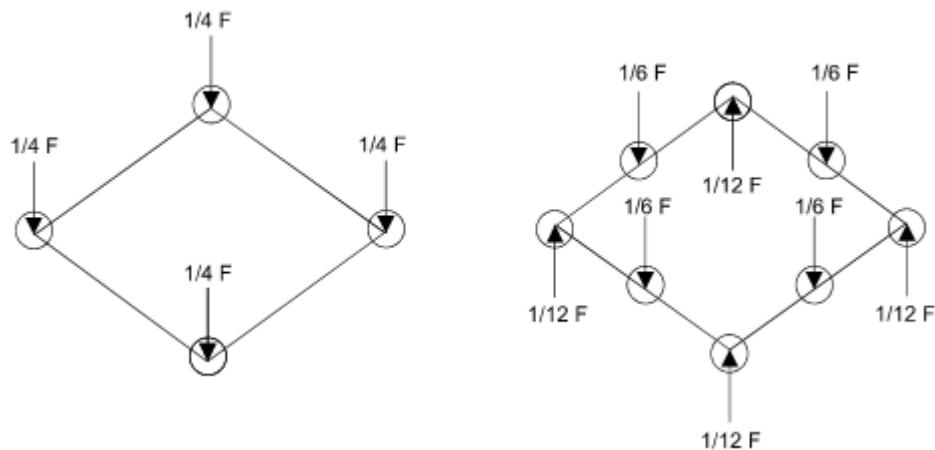
Example of **tetrahedron**, **pyramid**, **penta** and **hexa** elements

First Order	4, 5, 6, 8	nodes
Second Order	8, 12, 15, 20	nodes
DOFs	3	degrees of freedom per node

Higher Order Elements

Higher order elements are those with one or more mid-side nodes, or geometry based elements, such as p-version elements. These types of elements offer the benefits of ease of modeling and a higher degree of accuracy per element. P-version type elements also have a built-in ability to check convergence by increasing the integration level although it is more difficult to understand their fundamental behavior.

Higher order elements give rise to issues such as the sophisticated methods required to apply pressure to the face of a shell element. The required distribution of nodal loads to accomplish the same resultant force ($F=P \times A$) on a 4-node and an 8-node shell element is shown below.



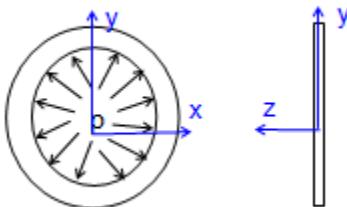
Consistent Pressure Loads For Shells ($F = P \times A$)

Most codes handle these details, but you should understand these and the other fundamentals of higher order elements to avoid confusion. Higher order elements are most often used in 3-D solid modeling because the potential to reduce modeling effort and the number of elements required to capture the geometry is greater. Solution time is not often reduced however because the global stiffness matrix is based on nodal DOF in the model.

Plane Stress And Strain Element

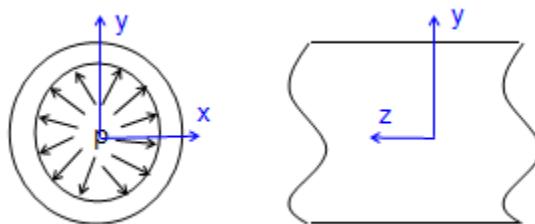
Plane stress: $\sigma_z = \tau_y = \tau_x = 0 \quad \varepsilon_z \neq 0$

A thin planar structure with constant thickness and loading within the plane of the structure for example:



Plane strain: $\varepsilon_z = \gamma_y = \gamma_x = 0 \quad \sigma_z \neq 0$

A long structure with a uniform cross section and transverse loading along its length, for example:



9.11 Element Selection

Element selection is based on the type of problem you want to run, boundary conditions, geometry considerations, and results required. Most problems can be solved many different ways and there is no “right” answer to the question of element selection, but making a good choice can reduce effort, computer time, and errors in the results. Often the solver you choose to solve the problem will have limitations for some element types and not for others restricting element selection.

Masses (0D elements)

Masses are point load masses that are generally used to represent attached structures at their centroids. This is an extremely good way to represent otherwise complex structures when the detailed is not required.

Beams & Rods (1D elements)

Types of 1D elements:				
Rod	Bar	Beam	Pipe	Axisymmetric shell
Tension/compression (and Torque for some software) U_x, R_x (1,4)	All 6 dofs $U_x, U_y, U_z, R_x, R_y, R_z$ (123456) Applicable for symmetric c/s	Same as bar but also support unsymmetric sections i.e. shear center and warpage	Same as beam. Except it has internal non zero diameter	U_x, U_z, R_y (1,3,5) Z-axis of symmetry, X as radial axis. For objects symmetric about the axis of rotation and subjected to the axisymmetric boundary conditions.
Tension compression members (truss), Shafts subjected to Torque, Connection elements	Shaft subjected to multiaxial loading, bolted, welded joints, connection elements	Same as bar + for unsymmetric c/s	Piping systems, Structural analysis	Thin shell pressure vessels, cylindrical, conical objects etc.

Beams are characterized by long and slender members, such as a space frame or a formula racing suspension. Bridge members are also good examples of beams or spars. Beams are very useful because of the flexibility in modeling complex cross-sections without modeling the geometry, but the burden of maintaining the detailed information is upon the user. In addition very accurate stress and deflection results are achievable with beam elements, but the visualization of the results is sometimes difficult.

Some examples of 1D elements are listed below:

- Rods
- Welds
- Rigids

Rods are essentially 2D beams and are great for in plane problems. Welds and rigids are used for defining constraint equations between nodes. Generally this results in an independent node and a dependent node(s) that form a set of equations that are placed in the stiffness matrix.

Plates (2D elements)

Plates are 2D elements that represent 3D space by assuming an infinite depth, fixed depth, or axisymmetric geometry. They have a reduced stiffness matrix and therefore reduced solution time with no loss in accuracy if the assumptions for the element hold.

Shells (2.5D elements)

Shells are essentially 2D elements that represent 3D space, thus the term 2.5D. Shells are excellent for thin 3D structures, such as body panels, sheet metal, injection molded plastic or any part that can be described as having a thickness that is small relative to its global dimensions. Deflections are given at the nodes, but stresses can be found at the upper and lower surfaces as well as at the midplane. This gives the analyst the ability to extract membrane effects versus bending effects in the results.

Solids (3D elements)

Solid elements are generally used for 3D structures not fitting into the shell description. Castings, forgings, blocky structures, and volumes are all good examples of 3D solid element structures. Solid elements have the benefit of eliminating many assumptions found in the other element types but are generally more difficult to model.

9.12 Mesh Density And Solution Convergence

Mesh density and solution convergence are closely related and the factors which determine that relationship can be controversial. In an effort to meet specific time and accuracy requirements, trade-offs involving modeling time, accuracy, computation time, and cost must be made.

The correct mesh, from a numerical accuracy standpoint, is one that yields no significant differences in the results when a mesh refinement is introduced. Although this concept may sound simplistic, many factors must be considered. Mesh refinements must accurately represent the problem in question if they are to be used in the analysis. Mesh refinements by simple splitting of elements can be misleading unless the newly created nodes conform closely to the original geometry. As refinement progresses, the original element selection must retain its significance. For example, a shell model can be refined to the point that it loses its validity in the area of interest, creating a need for a solid element model.

Determining a mesh density is facilitated by following a few basic guidelines.

Geometric Detail Required

Determine the smallest geometric detail(s) that must be captured in the model to obtain the results. A very sharp radius may cause a stress concentration, but at the same time, it may not be in a load bearing component of the assembly. The modeling required to capture this detail may require a separate local analysis after an analysis of the overall structure or component has been conducted.

Design Detail Available

Observe the degree of detail in the available design data. If the design data is preliminary or incomplete, or if you are using finite element analysis to help define the design, it is best to keep models simple. Take care not to oversimplify models to the point that factors under investigation are missed.

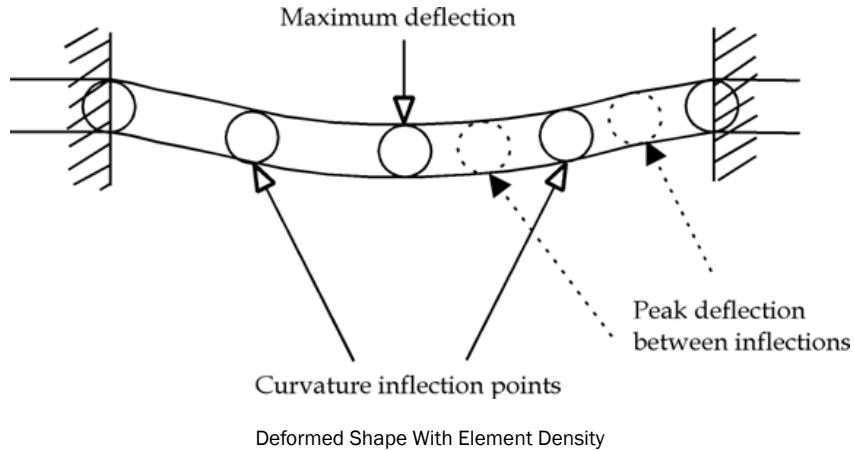
If the design is considered complete and a final verification is being conducted, include as much detail and mesh refinement (including re-mesh iterations for accuracy checks) as time permits.

Comparisons To Previous Work

If you will be comparing your work to other analysis results for the same or similar components, consider using a previously used mesh density which is similar. Consider correlations established with the testing of past models but be prepared to identify improper boundary conditions or load applications, poor modeling techniques, or inadequate mesh density. If such testing flaws are discovered, establish a new standard. Do not accept the work of others until it is fully understood.

Expected Deformed Shape

Determine possible deflection shapes and the mesh densities required to capture them. Estimate the maximum deflection areas and areas of curvature inflection. Observe whether nodal density follows the deflection pattern closely. Note the pattern shown below.

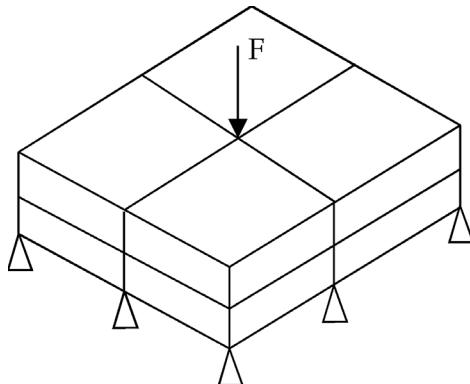


Checks Of Convergence

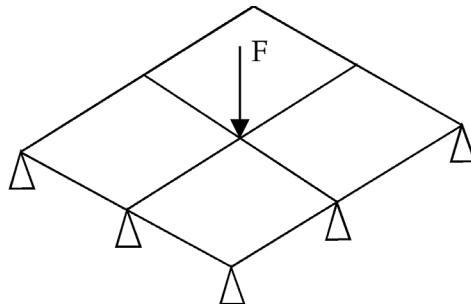
If you plan to do a convergence check, consider performing at least one refinement of the model after the first run. If neighboring elements display large differences in stress, the gradient was probably not captured in these areas, therefore some refinement is recommended.

New techniques for automatically computing convergence by several criteria are available on certain codes or can be customized by the user. Some computer codes will also automatically re-mesh nonconverged portions of a model. These techniques are dependent on the load cases and will provide different meshes for different load cases.

The p-version accomplishes refinement by numerically increasing the complexity of each element or nonconverged elements on subsequent re-runs. All automated techniques require that the model be set up to "near perfect" or the convergence may focus on small details that are irrelevant to the specific problem or have not been modeled with elements, boundary conditions or loads that have a natural converged solution. It is best to set up a problem so that there is a chance for convergence even if you do not intend to perform a convergence test. Below is an example of a flat structure that will converge if modeled in shells but will not converge if modeled "more accurately" with solids. Since the shells do not sense stress in the direction of the load, they will converge to the shell theory solution. The solids will attempt to resolve the "point load" and the stresses will go higher and higher as the elements under the load decrease in size.



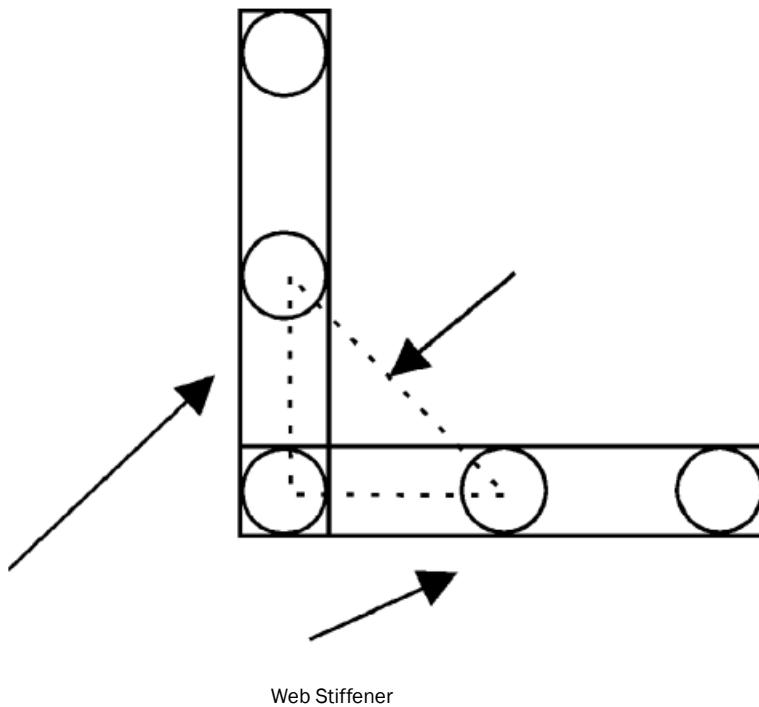
Solid Model
(will not converge)



Shell Model
(will converge)

Deflection Or Stress

In most cases, far fewer elements are necessary if only deflection or stiffness information is required. Even fewer elements can be used if only the deflection under the load is to be studied. For instance, one beam or one shell can estimate the deflection of a cantilever beam of rectangular section. It may, however, take dozens of shell or solid element to capture the stress at the boundary in a simple cantilever beam. If there is a high gradient of stress over a large area, for example, a web in the corner of a frame, many elements may be required to get the proper deflection.



Areas Of Concern

Determine the areas of concern. If you have a known issue in the lab or the field you will certainly want some detail in these areas. If time is limited, you may want to focus only in these areas and use rough approximations else where. If you have no prior knowledge of the component or system and a history search has not turned up any clues, you may have to make a very uniform mesh and refine areas that respond to the applied loads. Use your engineering sense to predict the areas that are prone to trouble. Some typical areas of concern are listed below:

- Tight radii
- Points of load application

- Boundary points
- Attachment points
- Narrow sections
- Abrupt change in section

Element Types

Mesh density is very dependent on the selection of element types therefore select the element type that is appropriate to solve your problem. Determine whether the structure should be considered as a shell type or if a full 3D model is needed to capture either the state of stress, geometry, or local deflections.

For example, engine blocks can be modeled with shells and beams when stiffness or dynamics is the only consideration. However, the stresses cannot be studied since this component is a 3-D solid when stress becomes a consideration. The time difference to model an engine in solids versus shells and beams can be enormous. Below it is a list of **Rules of Thumb** regarding element selection:

- Do NOT mix trias and quads whenever possible and if required place trias in areas of non-concern
- Use all trias for back-to-back comparisons when you have determined trias are adequate to your solution - don't compare a quad model with a refined tria model.
- Use elements of consistent size whenever possible
- Do NOT combine shells and solids or beams and shells/solids without first understanding all the assumptions and implications to your solution
- Model solid parts with solid elements
- Model thin plate structures with shell elements (thickness < 10-20 x Edge length)
- Understand all assumptions for the element type you have selected
- Do NOT use degenerated elements unless required
- Do NOT mix tetrahedral elements with other element types and, if required, place outside areas of concern
- Use tetrahedral elements when the effort required to model hexahedral elements is excessive (this can run into weeks of effort versus days for tetras)
- Use beam elements when shells or solids require excessive modeling time and effort

Beam elements can be used effectively in beam-like structures or for fasteners and connections. However, due to the complexity of employing beam elements, they are probably the most misused elements in the family of elements. The following considerations make using beam elements a difficult process to perform successfully.

- Shear center
- Warping constraint
- Length to depth ratio
- Shear deflection
- Complex state of stress near end conditions
- Visualizing both the input and the output of beam analysis

Stress results for complex beam sections are only good for global type values computed by classical beam methods. Beams of circular cross section are an exception because they are easily modeled and the classical beam results usually match actual performance except at localized end conditions.

Decades of study and research have enabled experts to construct intricate geometry with beam elements (usually to save

computer time) but few individuals understand the complexity of these techniques. Therefore be prepared to encounter a very involved situation if you choose to employ this element type.

Timing

Available time is a major determinant in a decision concerning mesh density. A compromise is usually reached between the amounts of time dedicated to human time and that devoted to computer time. It is often less time consuming to build a large model by using an automatic mesher, and increased computer capacity has lessened the need to use special techniques to minimize the number of elements. Therefore human time has become more focused on productivity and competitiveness in the market. Pick modeling techniques and elements that provide the most efficient blend of factors while simultaneously considering fixed external factors. Once you are convinced that your decisions are solidly supported, discuss your approach with others before implementing your plan.

Another important factor in the decision on model complexity is the tendency for models to “live” or an extended period of time. If you feel that the model may have a long life then work at ways to simplify the modification and/or redefinition to reduce future effort. This is an often overlooked problem and can cause significant time loss.

Mesh Density And Solution Convergence Summary

- Identify the smallest details that must be captured.
- Evaluate the level of design detail available.
- Check for current standards and previous work that will be used for comparison.
- Estimate the deformed shape and its requirements on your model.
- Plan for a convergence iteration(s).
- Select a more detailed modeling approach when stress is important. Consider a two-phase approach to stress solutions in which you make a second model for detailed work to reduce the scope and complexity of your solution.
- Identify known and predicted areas of concern.
- Select appropriate element type(s). Mix element types only with caution.
- Consider timing in your approach.

Recommended Video (not related to HyperWorks)

- **Convergence of finite elements**

(Prof. J. Chessa, University Texas; <http://youtu.be/YRJ0c8u08iU>)

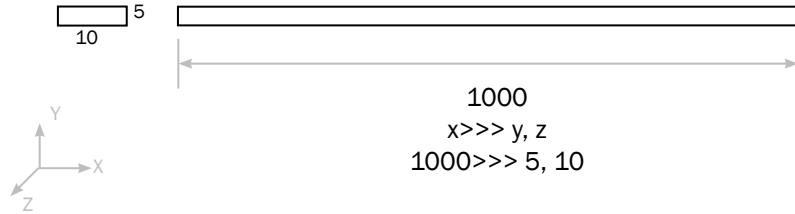


10 1D Meshing With HyperMesh

This chapter includes material from the book “**Practical Finite Element Analysis**”. It also has been reviewed and has additional material added by Matthias Goelke.

10.1 When To Use 1D Elements

Recall, we typically use 1D elements if one of the dimensions is very large in comparison to the other two:



Element shape – Line

Additional data from user - The remaining two dimensions, the cross sectional area.

Element type – Rod, bar, beam, pipe, axis-symmetric shell etc.

Practical applications - Long shafts, beams, pin joint, connection elements

In practice , the following element types are commonly used

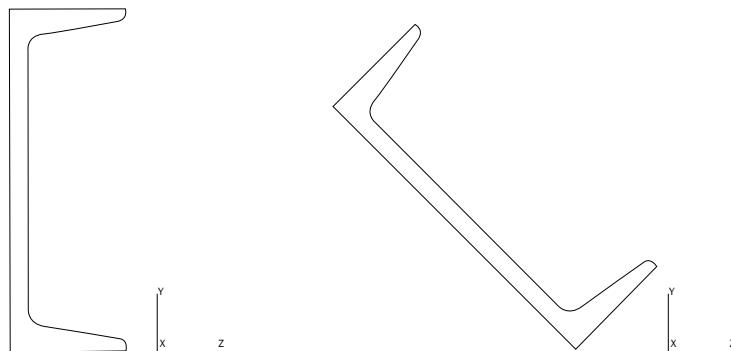
Types Of 1D Elements:

Rod	Bar	Beam	Pipe	Axisymmetric shell
Tension/ compression (and Torque for some software) $U_x, R_x (1,4)$	All 6 dofs $U_x, U_y, U_z, R_x, R_y, R_z (123456)$ Applicable for symmetric c/s	Same as bar but also support unsymmetric sections i.e. shear center and warpage	Same as beam. Except it has internal non zero diameter	$U_x, U_z, R_y (1,3,5)$ Z-axis of symmetry, X as radial axis. For objects symmetric about the axis of rotation and subjected to the axisymmetric boundary conditions.
Tension compression members (truss), Shafts subjected to Torque, Connection elements	Shaft subjected to multiaxial loading, bolted, welded joints, connection elements	Same as bar + for unsymmetric c/s	Piping systems, Structural analysis	Thin shell pressure vessels, cylindrical, conical objects etc.

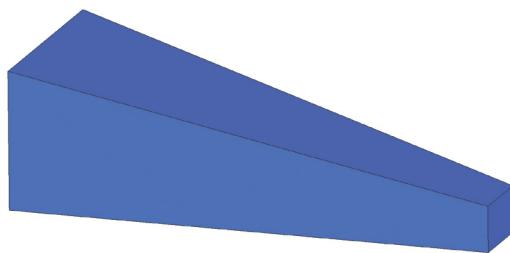
10.2 Special Features Of Beam/Bar Elements

a. Orientation of cross section

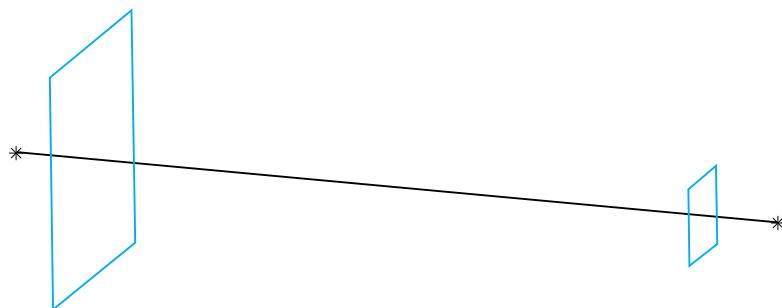
Changing the orientation angle of the cross section is supported by most commercial software and will be discussed in some detail further below.



b. Taper elements

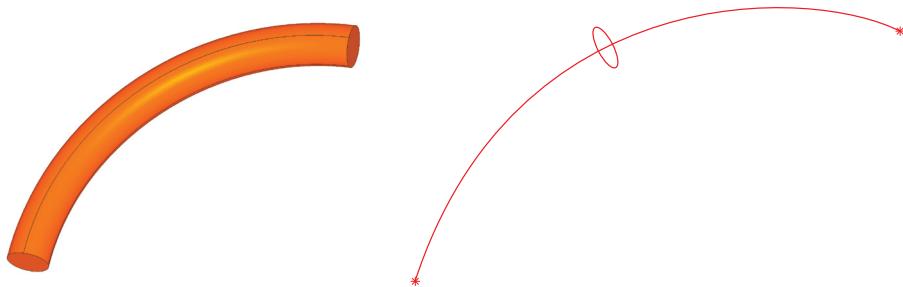


Regular bar elements cannot take into account a variation in the cross section. This could be achieved by using a taper beam. Input data for a taper beam is two cross sections (front and rear).

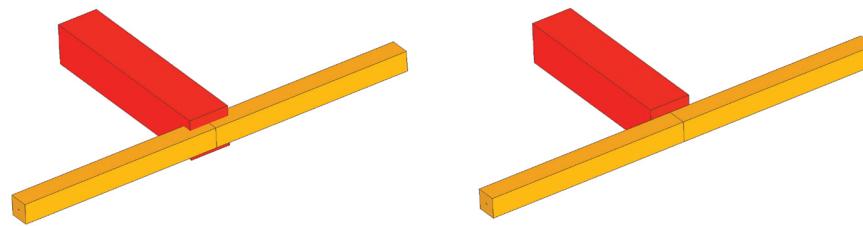
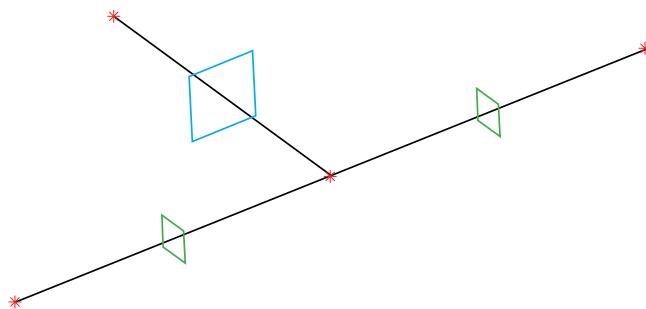


c. Curved beams:

Employed to model curvatures



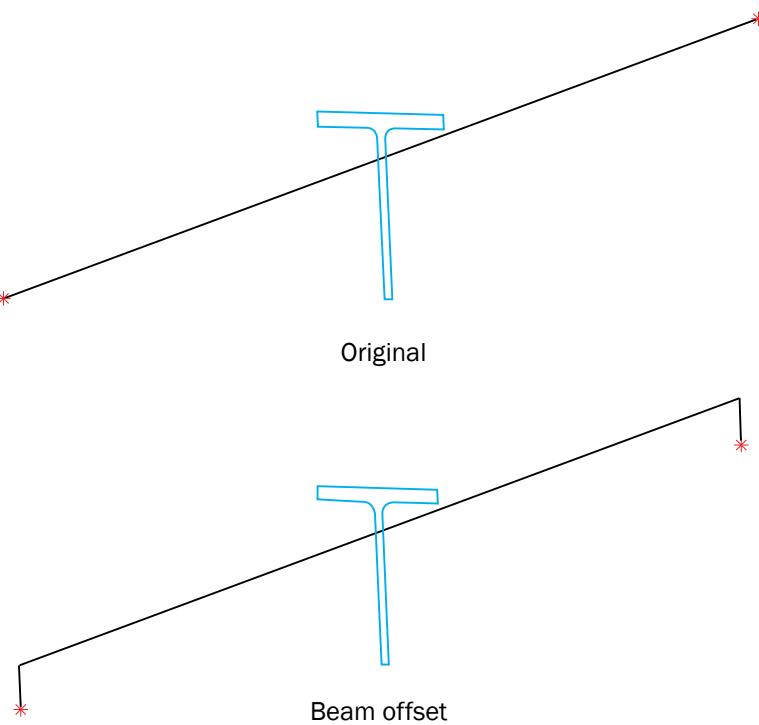
d. Beam end shortening: To represent the geometry accurately at joints



Without end shortening (interference at joint)

End shortening

e. Beam cross section offset: Useful in representing the geometry as per a real life situation



f. Beam end release

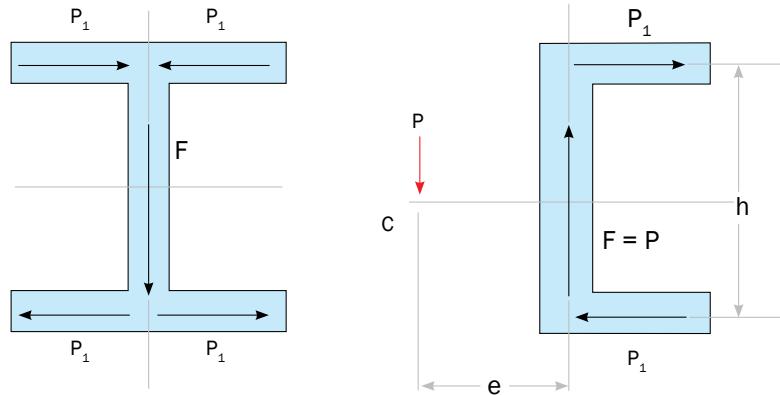
A beam element has 6 dofs. There are special situations, like a rotating shaft or hinge joint, where the rotation or translation in a specific direction is free. The use of a regular beam element and its spider connection, using Rigid or RBE2 elements, might not represent the exact situation. The end release is a special command to release one or more dofs of a beam.



For example, consider the door above. Its rotation about the vertical axis is free. Suppose the pin of the hinge is simulated by a beam element. The end release (rotation along vertical axis) would be helpful in simulating the door in real life.

g. Shear center

It is applicable only for non symmetric cross sections (like angle or channel cross sections). When a vertical shear force is applied to a symmetric cross sectional beam, it will bend. But for an unsymmetric cross section, the same force would result in bending as well twisting. Twisting is due to unbalanced shear forces in the flange. The shear center is the point at which when a force is applied, will nullify the effect of torsion.



For symmetric cross sections, the centroid and shear center coincide, therefore there is no twisting.

Shear flow for an I - section shows that the forces in the flange are in equilibrium and thus there is no twisting. While for a C channel, there is an unbalanced couple which gives rise to torsion.

The shear center is a property of the given cross section and it does not depend on the magnitude of the force.

Beam elements are recommended for symmetric as well unsymmetric cross sections, as it takes into account the shear center. Bar elements have a limitation on the shear center and therefore are not recommended for unsymmetric cross sections.

10.3 Rigid Elements

Rigid “elements” were devised to simplify the input of constraint equations for commonly used configurations. They are not structural elements, but are equations that define dependencies between degrees of freedom. There are many illegal configurations of rigid elements which lead to a large percentage of input errors therefore they should be used with care. Because there really are no perfectly rigid materials in nature, a rigid element is usually a poor representation of any structural component. In general, a rigid is simply a constraint equation or a set of constraint equations of the form:

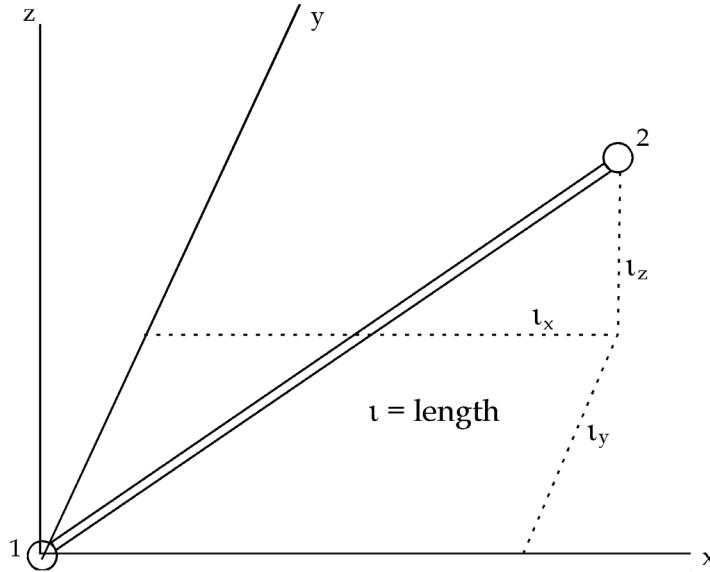
$$A_1 u_1 + A_2 u_2 + A_3 u_3 \dots = 0;$$

where the A = constant coefficients and the u = degrees of freedom. Usually, one degree of freedom is dependent on all of the others.

RBAR Element

The RBAR element rigidly connects from one to six dependent degrees of freedom (the m -set) to exactly six independent degrees of freedom. The six independent degrees of freedom must be capable of describing the rigid body properties of the element.

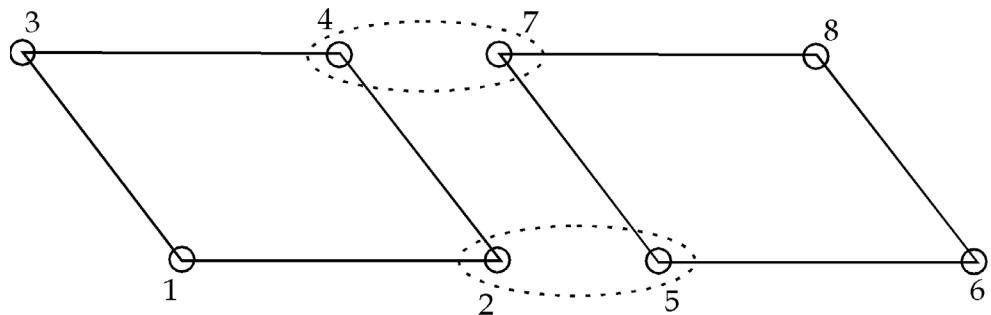
If a rigid element is long or of similar length as the structure and is strongly coupled to the structure, it is probably overly stiffening the structure. In this case a simplified representation using structural elements would be preferred. Using rigid elements for fasteners such as bolts typically results in overly stiff joints. The use of a beam for modeling a bolt not only gives a better representation but allows easy output of force quantities.



Proper Application Of Rigid “Elements”

When part of a structure is very stiff relative to and lightly coupled to, the structure under investigation, it is a candidate for application of rigid elements. An example might be an engine connected through soft mounts to a frame. The engine might be considered rigid if its deflections have little effect on the frame.

Rigid elements are also appropriate when modeling a mechanism, such as a hinge as part of a structure. In this case, two nodes are placed at the same location and only coupled degrees of freedom are constrained together. In fact, this type of zero length rigid element is a very important application of rigid element. See the following figure depicting a hinge.

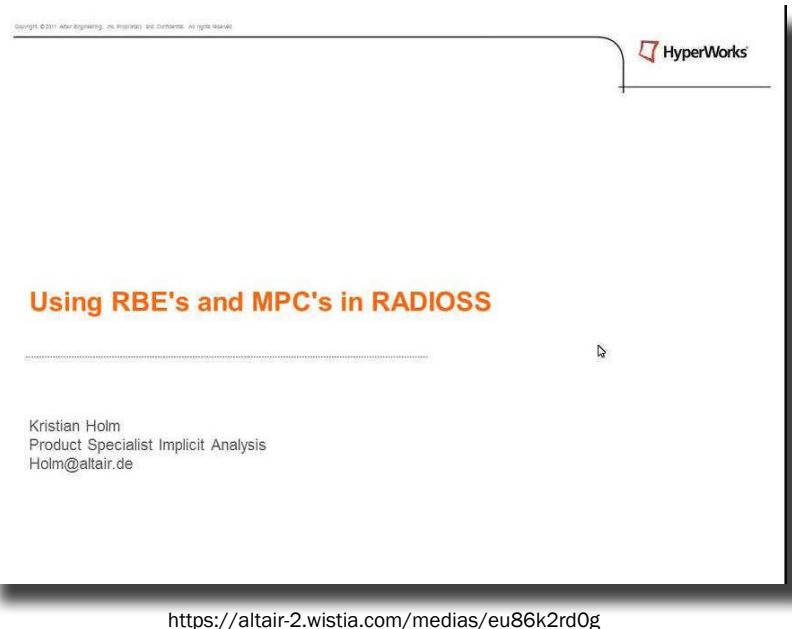


Pairs 4 & 7 and 2 & 5 are modeled at the same location and constrained in all degrees of freedom except rotation around the y axis.

Hinge using zero length rigid elements

Webinar On RBE And MPC Elements

We recorded a webinar about how to use RBE and MPC elements. Likely errors and their impact on modeling results are discussed. In addition, tips are given on how to avoid such errors.



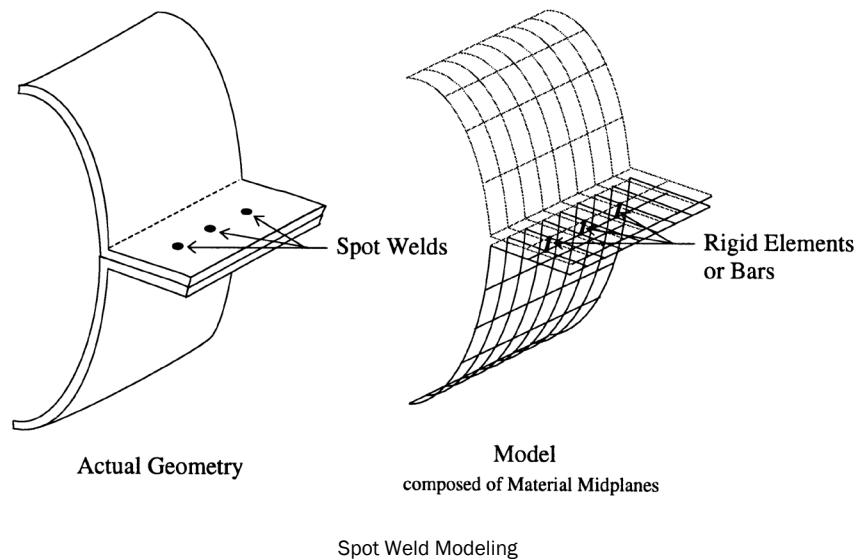
10.4 Fasteners

Modeling of fasteners and welds using the finite element method is an arguable procedure. The details around a fastened joint are complex and often exhibit nonlinear behavior. The best approach to adopt is one in which the techniques give an indication of good versus poor attachment. An “absolute” evaluation of fastener performance would be extremely difficult using current standard methods. The following section will discuss some of the methods used in modeling for a variety of connections.

Spot Welds

Spot weld connections are normally used between relatively thin sheet metal components such as those used in car body design. Rigid elements or common nodes (for very thin members) are used to model spot welds. Care must be taken to ensure that each rigid element is normal to the surfaces it attaches. It should be noted that the lack of “drilling” degrees of freedom in some solvers will allow the welds to virtually spin in flat areas of the structure causing matrix singularities. Automatic singularity elimination codes will usually ignore nodes attached to rigid elements, so constraints will need to be added manually or a soft rotational spring will be added in the normal direction of the element. This approach is sometimes referred to as fictitious “drilling” stiffness option.

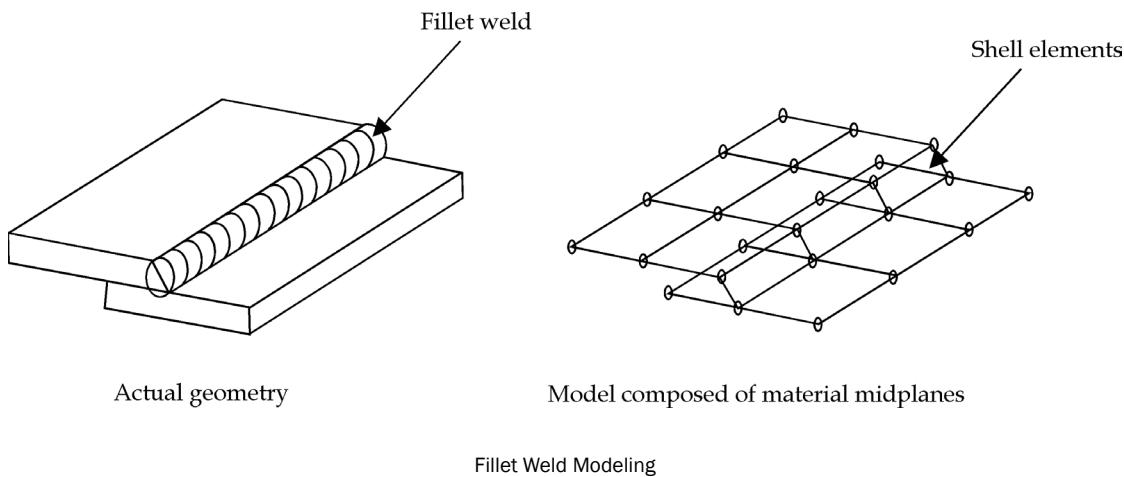
If beam elements are used instead of rigid elements, the singularities will be found and automatically eliminated. A secondary benefit of using beams for spot welds involves the ease of recovering forces in the welds. A round beam with the approximate diameter of the spot weld and parent material properties would work well as long as it had a non-zero length. The figure below illustrates a typical modeling scenario for spot welds.



There should be at least one node on the parts between spot weld nodes to allow relative motion between the two parts to be joined. If there are no degrees of freedom between neighboring spot welds, then there is, in effect, a continuous seam weld simulated. The forces recovered in this type of model can be compared to test data on similar welds to determine if the number and placement of welds will be adequate.

Fillet Weld

Fillet welds normally occur between heavy gage plates or stampings. These welds can be effectively modeled by connecting shell elements if the parts are also modeled with shell elements. Care must be taken to avoid excessive aspect ratios. The stresses will not be accurate but will give relative nominal values. The irregular geometry of the actual weld is too complex to capture in a finite element model. The nominal values can be correlated to test results or scaled up using notch sensitivity tables for the material in question. The following figure depicts this method of modeling.



It is also common to use rigid elements similar to spot welds for fillet welds.

Rivets And Bolts

To model riveted or bolted joints with a linear code, we must assume that the joined surfaces either will never slip relative to each other or that they will always be able to slip. In most cases, it is assumed that the heads do not slip but the interface between the joined surfaces is allowed to slip. This assumption requires that the fastener carry the full shear load between the surfaces (conservative assumption), but that there are no prying forces in the joint (usually non-conservative assumption). It is best to design joints so that there is no slipping under the peak loads. This can be checked by looking at the shear force resultant and

comparing it to the intended installation tension multiplied by a conservative (lowest) friction coefficient. These methods require the application of basic engineering principals and the answers are not printed in the finite element output.

Pre-Loaded Joints

To properly account for the pre-load in a joint, the stiffness of the joint in the direction of the fastener axis must be included. This will normally require the construction of a detailed solid model of at least the joint area. The fastener can still be modeled with beams or also be represented with a full solid model. The pre-load can then be applied by a thermal shrink of the fastener or by application of equal and opposite forces on a segment of the fastener. The thermal or opposite forces will need to be adjusted to achieve the desired pre-load after an initial run determines the joint to fastener stiffness ratio. Usually this amount of detail is not merited because slipping during bolt up and under load, and prying forces are usually ignored. If the full details of a joint are to be studied a nonlinear code would probably be required. Today there are solvers that permit automatically apply the pre-tension load as a force or a prescribed displacement.

10.5 1D Element Creation

1D elements perform a critical function in Finite Element Analysis as they can be used to connect nodes together, attach dissimilar meshes, distribute loads and in general provide a quick and easy way to attach things together.

There are numerous types of 1D elements ranging from infinitely rigid simple connections to complex cross sectioned elements that can be stressed.

HyperMesh currently supports bar2, bar3, rigid links, rbe3, plot, rigid, rod, spring, weld, gap, and joint one dimensional elements. The following is a list of the different element types and their purpose.

- *bar2*: supports complex beams
- *bar3*: supports complex beams (bar3 elements contain a third node designed to support second order beams)
- *gap*: supports gap elements
- *joint*: supports kinematic joint definitions supplied with safety analysis codes
- *plot*: supports display type elements
- *rbe3*: supports NASTRAN RBE3 elements
- *rigid link*: supports rigid elements with multiple nodes
- *rigid*: supports rigid elements
- *rod*: supports simple beams
- *spring*: supports springs or damper
- *weld*: supports weld elements

HyperMesh also uses an entity called a **connector** which defines how different components or assemblies in the model are fastened to each other. These connectors are used to create or realize one-dimensional elements in the model.

Some Modeling Hints Regarding 1D Elements With HyperMesh

While working with 1D elements, sooner or later you will come across the following element types:

- CROD
- CBAR
- CBEAM

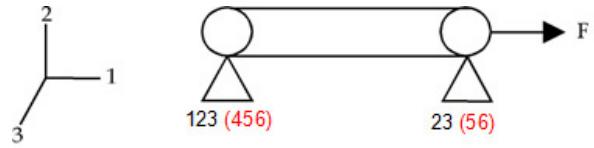
In brief, CROD elements support (allow) tension and compression only, whereas CBARS and CBEAMS allow bending as well. A CBAR element is a kind of simplified CBEAM element and should be used whenever the cross-section of the structure and its properties is constant and symmetrical. The CBAR element requires that its shear center and the neutral axis coincide. Due to this requirement, CBAR elements are not useful for modeling structures that may warp, such as open channel-sections. This limitation is not present in CBEAM elements.

Thus, CBEAM elements are used to model more “complicated” geometries with varying cross sections. Important to note, in CBEAM elements the neutral axis and shear center are noncoincident. Overall, this type of element demands a deeper understanding of beam theory.

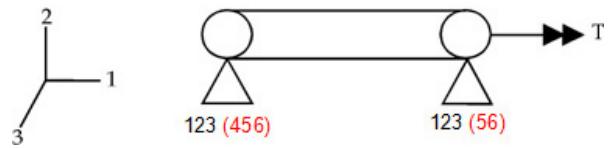
Furthermore, we recommend the beginner to use the predefined 1D element cross-sections available with the OptiStruct library (provided you are using OptiStruct as your finite element solver). Another benefit of employing the cross-sections available with the OptiStruct library is that the stress recovery points which are needed for postprocessing stress results are predefined already. Of course, in addition to the cross-sections depicted further below (OptiStruct library) you can create your very own and general 1D cross-sections with HyperMesh.

CROD Elements

In the following the general modeling building process based on CROD elements is documented. Most of the working steps can also be applied to CBAR and CBEAM elements, too.



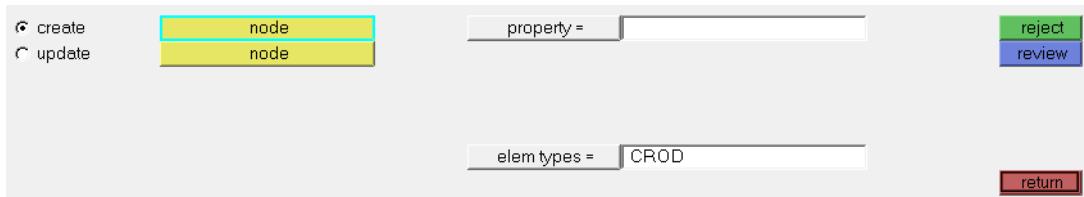
This element “understands” tension and compression loads (axial forces) only. In other words, the nodes of a CROD element only have translation degrees of freedom (still this element has a torsional stiffness).



How To Model CROD's

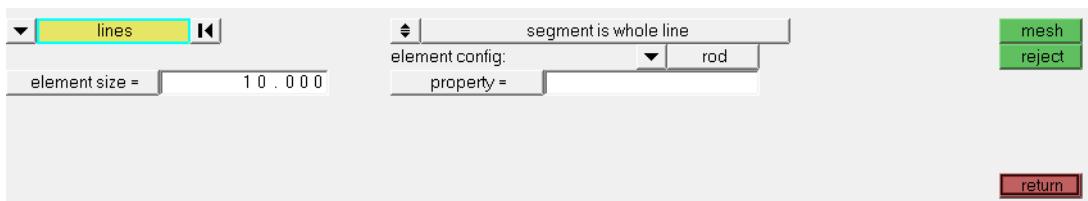
As we are going to use the FEM program OptiStruct, make sure that the corresponding User Profile is activated.

You may start by building CROD elements manually (with/without any property definition) by making use of the Rods panel. This panel is found in the menu bar by selecting Mesh > Create > 1D Elements > Rods.



Make sure that the “element type” to be created is set to CROD. In the simplest case, a CROD element is created by selecting its bounding nodes.

Other more general options to create many CROD's in a single step exist of course. For instance, by making use of the Line Mesh panel. This panel can be found through the menu bar by selecting Mesh > Create > Line Mesh.



Again make sure and specify the element configuration to be rod.

It may happen that the newly created elements are displayed as CWELD elements (not what we want).

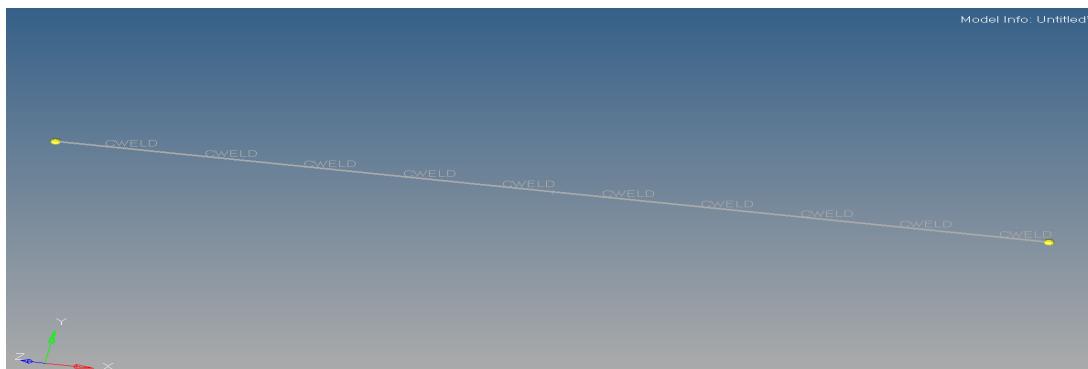


If this is the case, then the “element type” must be updated. This is done using the Element Type panel, which is found through the menu bar by selecting Mesh > Assign > Element Type.



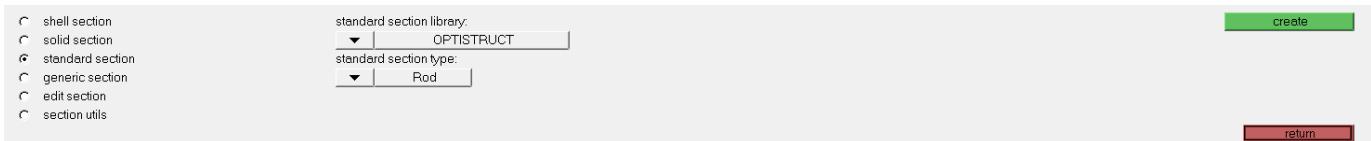
The element types are updated by setting rod= to CROD, then selecting the corresponding elements and the selecting update.

Up until now the element specific information, such as cross-section data, is missing. The CROD elements are thus simply displayed as a “line”:



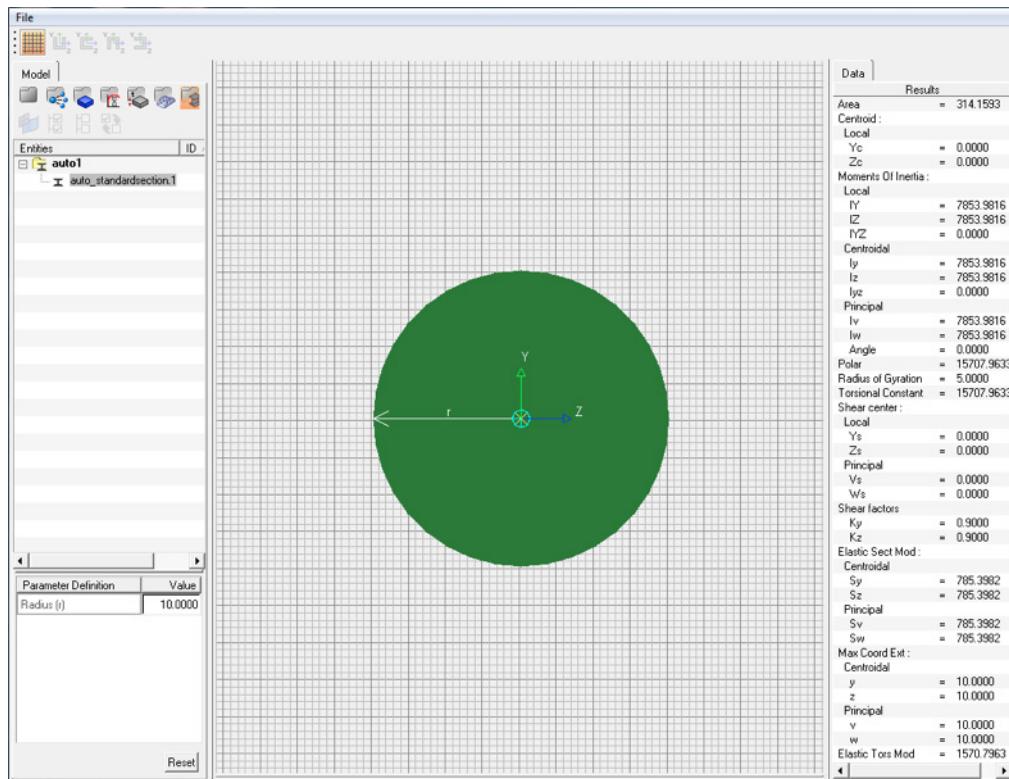
In order to define the elements cross-sectional properties, we recommend using HyperBeam which is found in the menu bar by selecting Properties > HyperBeam (also accessible from the 1D page > HyperBeam).

In the HyperBeam panel, select the standard section subpanel:



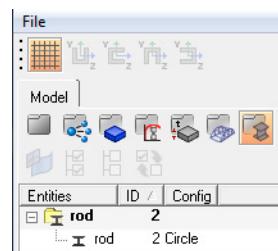
As stated before, we will make use of the readily available cross sections in OptiStruct. Therefore, make sure that for standard section library, OPTISTRUCT is activated. From the list of available cross-sections (standard section type:), chose the one of interest. Above it is set to Rod (Box, H at etc. are also valid of course). Note, the cross section is named "Rod" – it doesn't say anything about the 1D element type!

Clicking the create button then starts up HyperBeam which allows to interactively changing the radius of the rod cross-section.



Note, that all the relevant cross-sectional data such as area, moments of inertia are automatically calculated and listed in the side bar of HyperBeam. This information is stored as a beamsection collector (here named rod; listed in the side bar on the left). Later the beamsection will be referenced inside the mandatory property collector. Complicated? Not really, as you will see ...

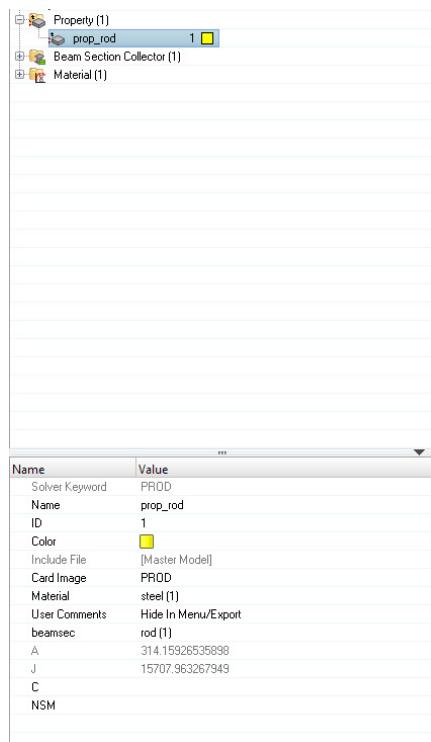
In order to switch back to the HyperMesh GUI, just activate the icon in the upper left menu bar (beneath Model in the Model Browser).



Before we proceed creating a property collector, we need to define a material (right mouse click in the Model Browser > Create >

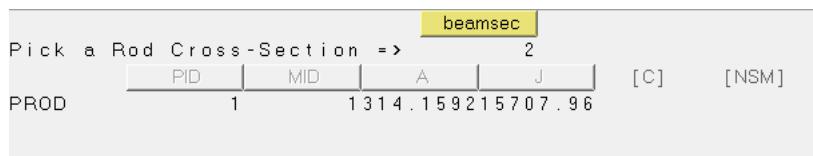
Material). In many cases, especially while getting started with FEM, the material card image will be MAT1 (MAT1 describes linear elastic isotropic temperature independent material behavior). The material is named steel using standard values for Young's Modulus, Poisson's ratio and density.

Then, in the next step the property collector for the CROD elements is created (right mouse click in the Model Browser > Create > Property). In the pop-up window of the property collector, the Card Image is chosen as PROD. In here we also reference the previously defined cross-section, namely the beamsection rod and the material steel.



Note, if you are using CBEAM or CBAR elements, the property Card Image would either be PBEAML or PBARL (L = as we are using the Library). Activate the Material tab next to Material, and assign the previously created material steel to this property collector (here the material is named "steel" (image above).

Eventually, the property collector is created. Its card image (solver syntax) then reads as

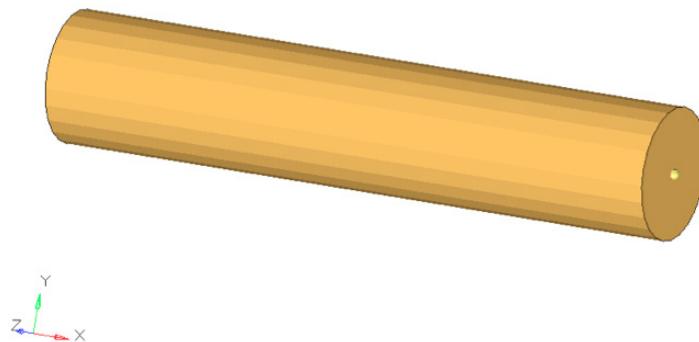


Where PID = property collector ID, MID = material collector, A is the area and J is the torsional constant.

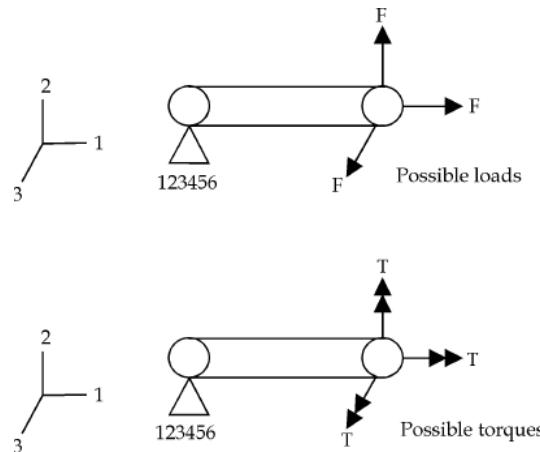
In the general analysis work flow (not restricted to 1D elements), the property collector must now be assigned to the elements of interest.



To visually check the attributes of the 1D elements, make sure that Shaded Elements and 3D Element Representation is activated. Then the CROD element will be displayed as:

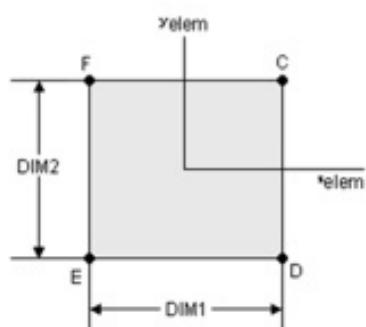


CBAR Elements

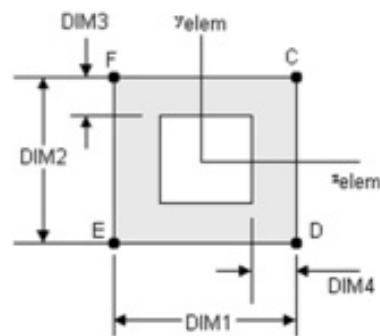


Simple beams with constant properties (symmetrical and constant cross-section), may be modeled with CBAR or CBEAM elements. However, what you need to recall is that in CBAR elements, the origin of the element coordinate system is centered at the shear center of the cross-section (shear center and neutral axis coincide). Any offsets between the neutral axis and the shear center is not accounted for. As a consequence, CBAR elements are not useful for modeling beams that warp, as may be the case with open channel sections. This is because the cross-sections of CBAR elements remain plane. In other words, whenever you are modeling open or nonsymmetrical sections, be especially cautious with the element type you are going to use.

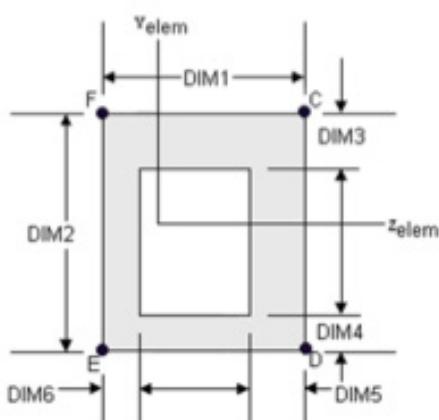
The following cross-sections shown below are available:



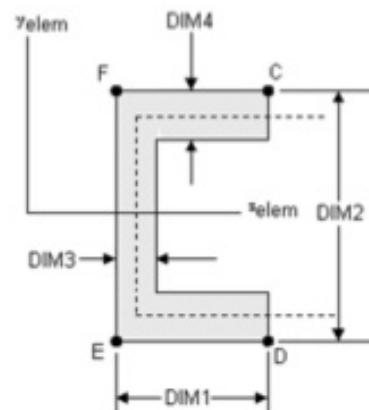
Type = BAR



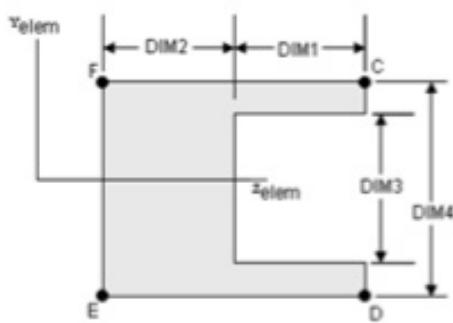
Type = BOX



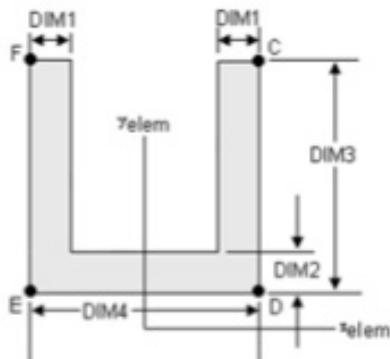
Type = BOX1



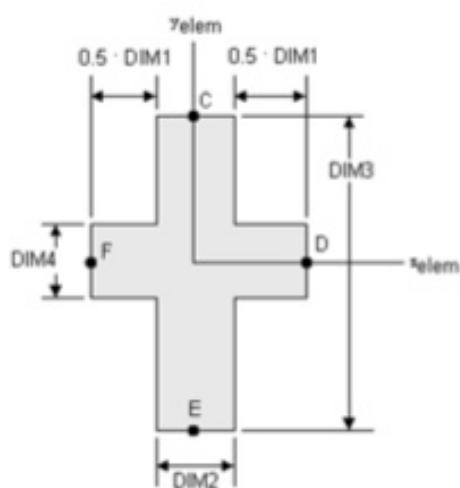
Type = CHAN



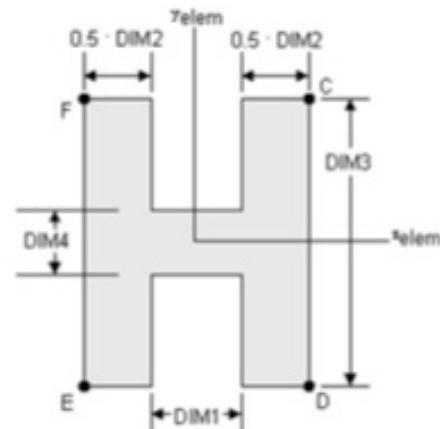
Type = CHAN1



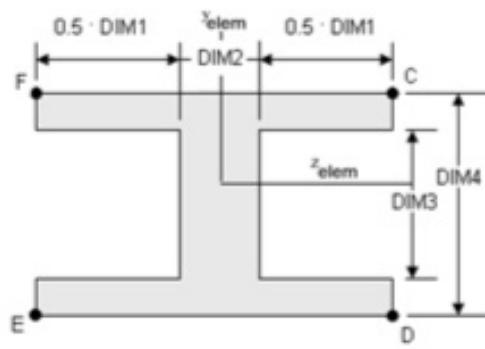
Type = CHAN2



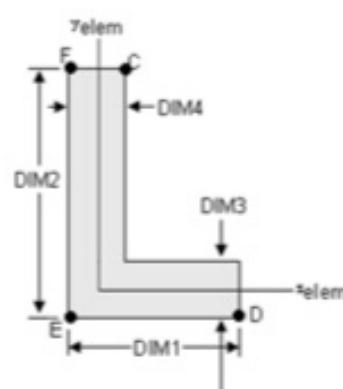
Type = CROSS



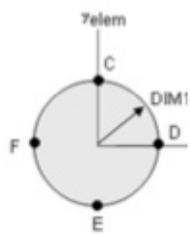
Type = H



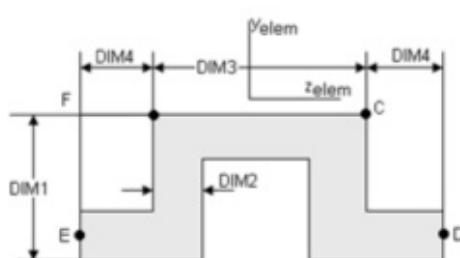
Type = I1



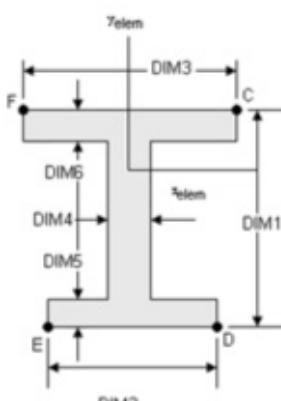
Type = L



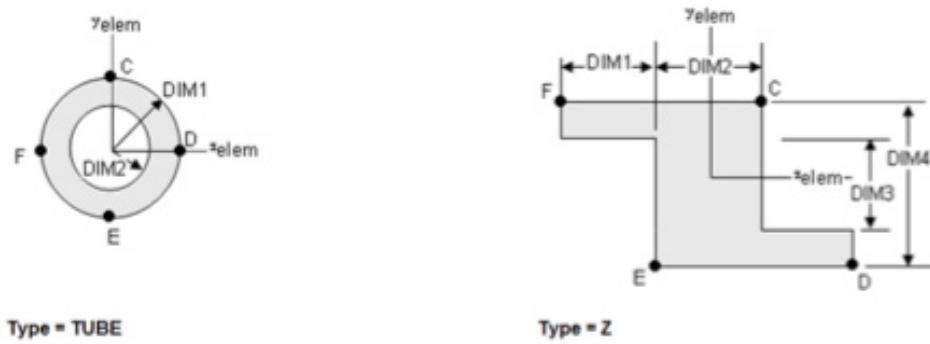
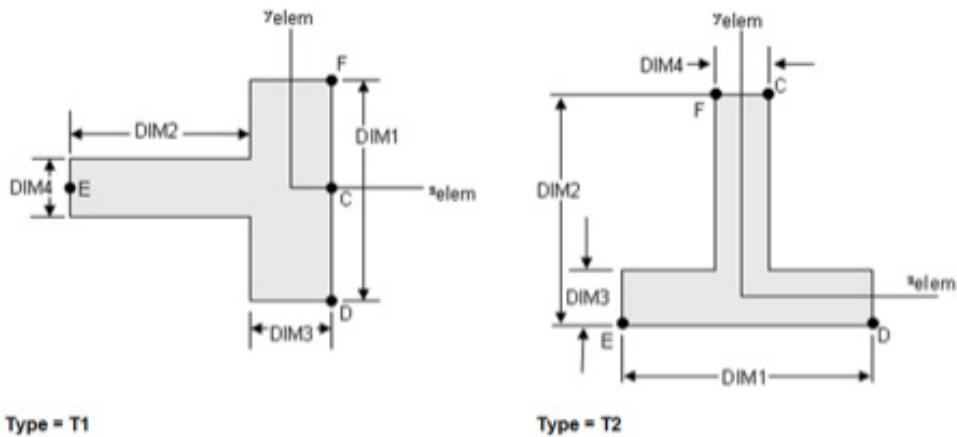
Type = ROD



Type = HAT

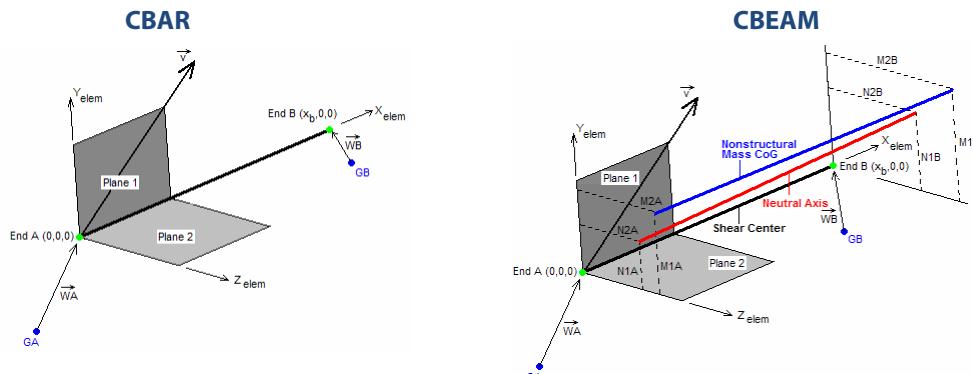


Type = I

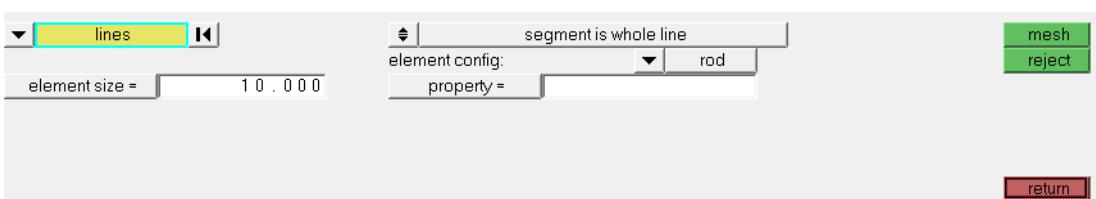


The cross-sectional properties, shear flexibility factors, and stress recovery points (C, D, E, and F) are then computed using the TYPE and DIMi as shown below.

As the CBAR and CBEAM elements also account for bending, the orientation of the element and thus of its cross-section becomes important (as it automatically defines the location of the pre-defined stress recovery points; see discussion below).

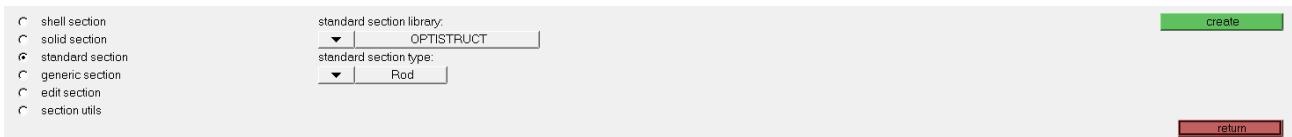


Meshing (building) of CBAR or CBEAM elements is just as simple as with the creation of CROD elements (e.g. Line Mesh panel)



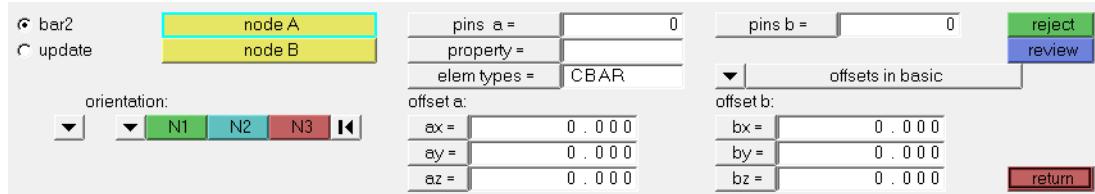
This time the element config is set to bar2 (if you miss this step, you still can update the element configuration later).

In order to define the cross-section, we again make use of HyperBeam and the OptiStruct library.



1. Building (manually) CBAR / CBEAM elements:

Start the meshing process with the Bars panel (Mesh > Create > 1D Elements > Bars)

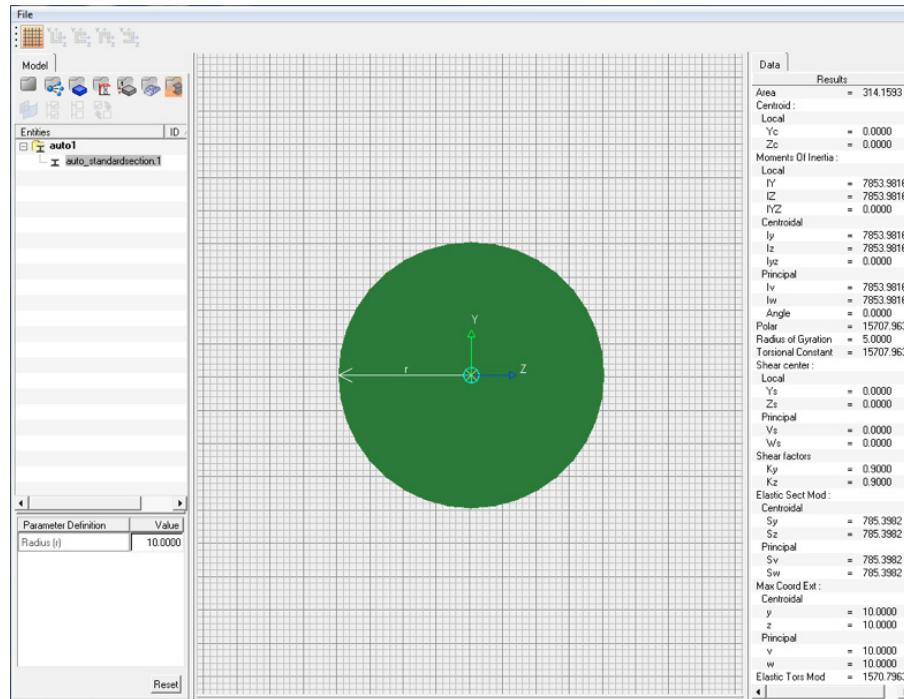


The element type is set to CBAR.

2. Specify cross sectional properties (with HyperBeam)



The standard section library OPTISTRUCT is used.

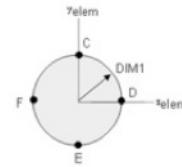
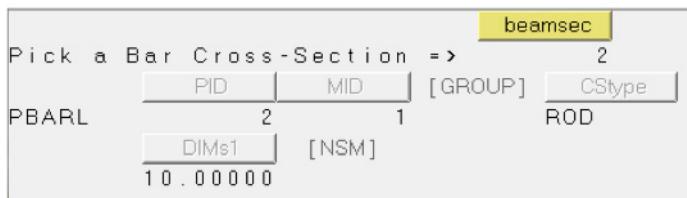


Again, the cross section properties can be interactively changed in HyperBeam. The beamsection is named rod_library and the beamsection collector is named rod

3. In the property collector, the Card Image PBARL is chosen i.e. L indicates that we reference the cross section from the library. Recall that the material named "steel" was created before ...

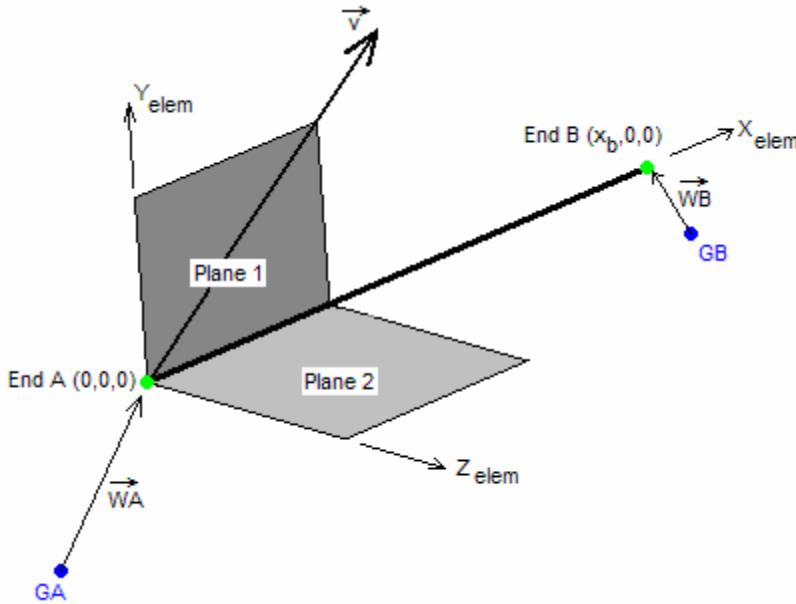
Name	Value
Solver Keyword	PBARL
Name	prop_rod
ID	1
Color	Yellow
Include File	[Master Model]
Card Image	PBARL
Material	steel (1)
User Comments	Hide In Menu/Export
Beam Section	Rod_library (1)
GROUP	
CType	ROD

The Card Image of the property collector now reads as:



Please compare this card image with the one belonging to PROD (from the example above). Here, the type of the cross-section (Ctype) is ROD as defined in HyperBeam, DIM1 corresponds to the radius of the rod, PID is the ID of the property collector, MID is the ID of the material collector, and beamsec references the before created cross-section.

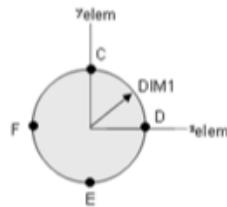
4. As mentioned earlier, care must now be taken regarding the orientation of the cross-section (orientation of the element and thus "location" of the stress recovery points).



The local element x-direction is given by its first (A) and second node (B). Its overall orientation, i.e. the local xy plane (Plane 1) of the element, is then defined by a vector v and the local x-axis. The v vector is defined with respect to the global coordinate system, however. Note, more information is also available in the help documentation (\help\hwsolvers\hwsolvers.htm, then Index > CBEAM or CBAR). To review and update the elements orientation (if needed) use the Bars panel (Mesh > Edit > 1D Elements > Bars).

In this example, the v vector parallels the global y-direction i.e. $ycomp=1$. Consequently, the stress recovery point C is located at the "upper" side of the rod.

A note regarding the stress recovery points: Its location essentially defines the c in the equation $\sigma = Mc/I$:



M = Moment, c distance from neutral fiber (i.e. location of the stress recovery point), I = Moment of Inertia. Based on c , Optistruct calculates the stresses in the rod or on its surface. Actually, the stress recovery points are automatically calculated at specific locations to give the maximum stress for the cross-section.

Visualization Of Element Stresses

Evaluating element stresses opens up a very different aspect. In the case of CBAR (CBEAM) elements, stresses are determined with respect to the stress recovery points C, D, E, F (see explanations above). This stress value is then used to contour the entire CBAR/CBEAM element.

In HyperView, node A or node B and the stress recovery point of interest must be specified, e.g. longitudinal stress at node A and recovery point C.

What Is The Meaning Of Pins a, Pins b, Offset a And Offset b?

So far the Bars panel was used to manually create CBAR (or CBEAM) elements.

bar2	node A	node B	pins a = 0	pins b = 0	reject
update			property = p r o p _ r o	offsets in displacement	review
			elem types = C B E A M		
			offset a:		
			ax = 0 . 0 0 0		
			ay = 0 . 0 0 0		
			az = 0 . 0 0 0		
			offset b:		
			bx = 0 . 0 0 0		
			by = 0 . 0 0 0		
			bz = 0 . 0 0 0		
					return

pins a and pins b

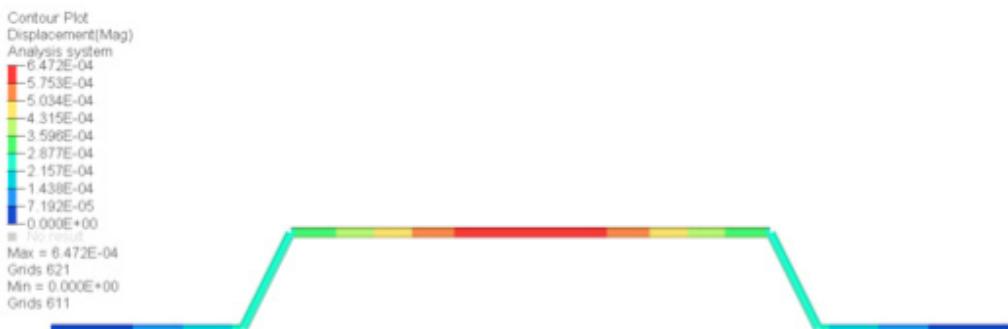
You may “pin” certain translations or rotations (degrees of freedom) of the 1- D element. In other words, if the 1D element is supposed to “transmit” all translations and rotations, then “pins a” and “pins b” must be set to 0 (zero). In case a certain degree of freedom must be “eliminated”, then the corresponding pin at node a or node b (eventually even both) needs to be pinned.

As an example, rotation with respect to the x-axis (dof 4) should not be transferred from node b to the attached element, then pins b=4. This still enables the 1D element to rotate around its x-axis, BUT as requested, this rotation is not transmitted to the neighbor element attached to node B.



offset a, offset b

Because bar elements are one-dimensional representations of three-dimensional objects, these values represent the distance between the real-world edge of a bar and its center axis. For example, if the bar was cylindrical in shape, node a and node b would determine where the edge of the bar is attached to another element. The offsets would identify where the bar's central axis lay in relation to the connection points.

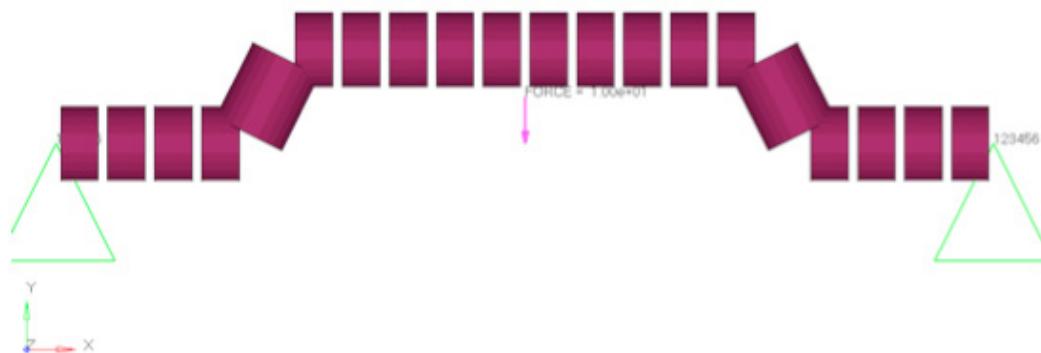


Note, in this model, the 1D elements physically follow the given geometry.

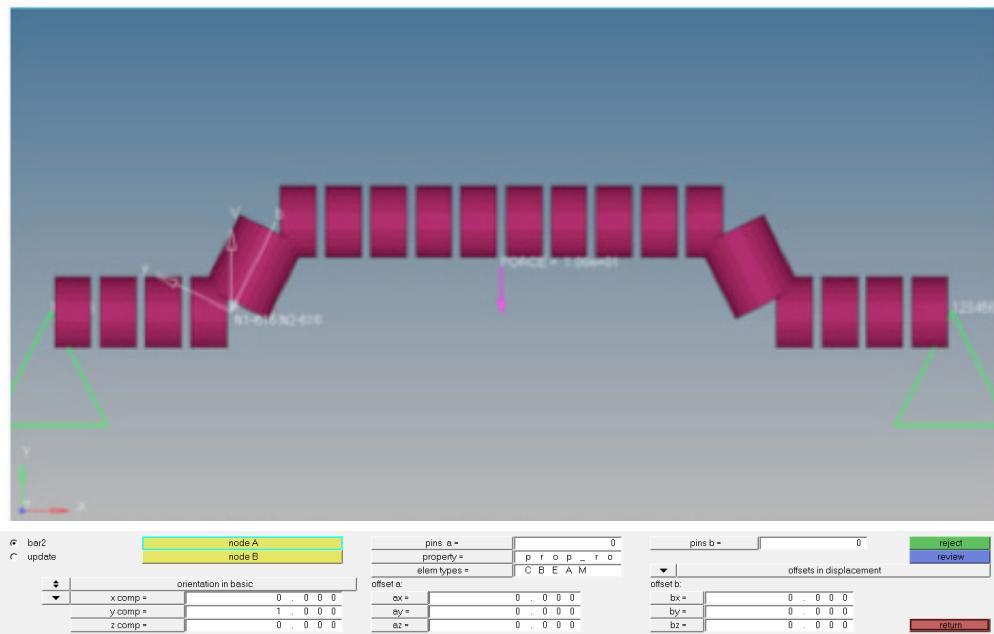
In the model shown next, the 1D elements form a straight line (elements are displayed with Shrink Mode and Traditional Element Representation .



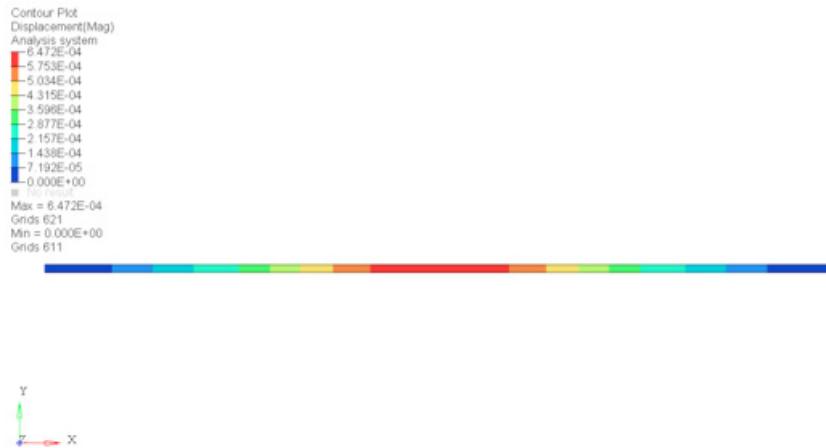
However, “internally” the elements nodes a and b are offset in order to account for the “true” geometry. This offset can only be visualized by activating the 3D Element Representation .



Reviewing the elements provides insight on how the offset is defined:



The offset of node B of the selected element is 4 mm (hence by= 4). This way, the geometry from the reference model shown above can be modeled without physically translating the elements.



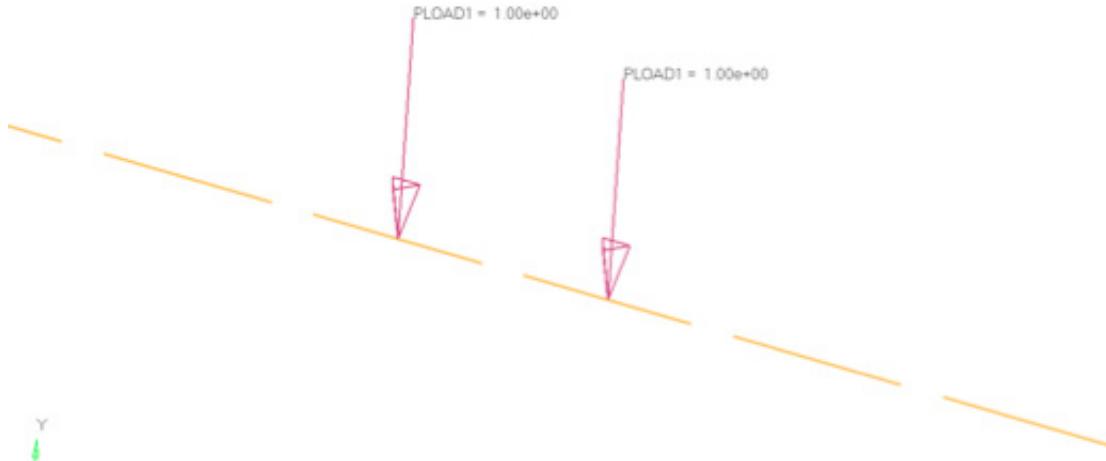
In the image above the displacement contour plot is shown. In this model the elements (i.e. nodes) are offset in order to account for the “true” geometry.

How To Apply Pressure On 1D Elements?

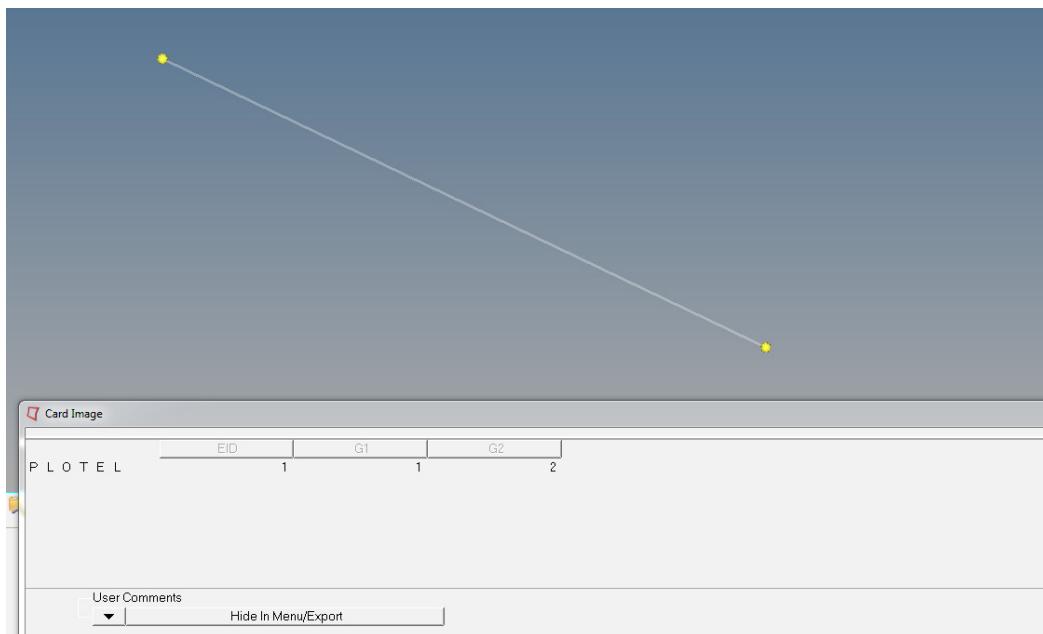
Pressure on CBAR or CBEAM elements may be applied as load type PLOAD1, which defines concentrated, uniformly distributed, or linearly distributed applied loads to the CBAR or CBEAM elements at user-chosen points along the axis (for detailed information see the help documentation help\hwsolvers\hwsolvers.htm > Pload1)

The load type PLOAD1 is specified in the pressure panel directly

<input checked="" type="radio"/> create	elems	uniform size =	2 . 0 0 0	<input type="button" value="create"/>
<input type="radio"/> update		<input checked="" type="checkbox"/> label loads		<input type="button" value="create/edit"/>
<input type="radio"/>	magnitude =	- 1 . 0 0 0 e + 0 0	<input type="button" value="reject"/>	
	y-axis		<input type="button" value="review"/>	
<input type="radio"/>	nodes on face:	nodes	<input type="button" value="return"/>	
	break angle =	3 0 . 0 0 0		
	load types =	P L O A D 1		



How To Convert PLOTEL Elements (General 1D Elements) Into CROD Elements



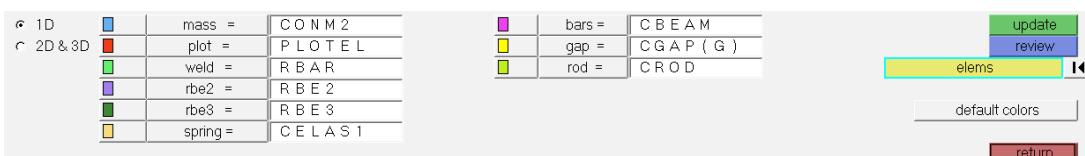
Note, the card edit option was used to display the element information.

Note, a meaningful material and property collector must exist. As we are converting the PLOTEL element into a CROD element, the Card Image of the property collector must be PROD. Consequently, if the conversion would be from PLOTEL to CBEAM or CBAR, the property collector Card Image should be PBEAM/PBEAML or PBAR/PBARTL.

1. Open the panel Element Config (Mesh > Assign > Element Config):

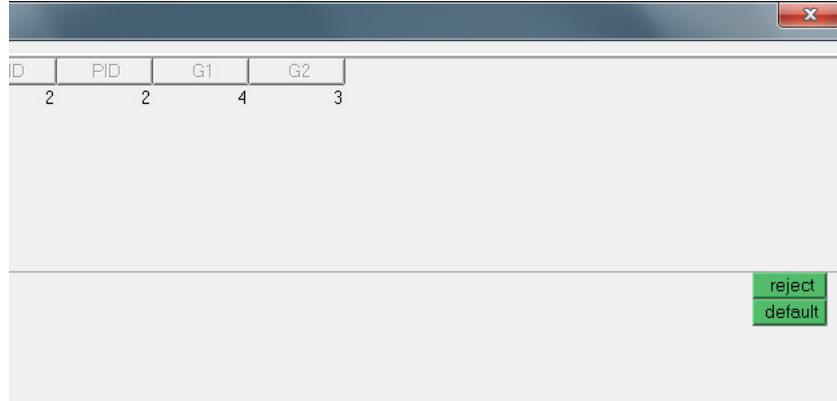
Specify the new element configuration e.g. rod (for CROD elements; bar2 for CBEAM/CBAR elements) and confirm the changes with switch.

2. Check the element types setting using the Element Type panel (Mesh > Assign > Element Type).



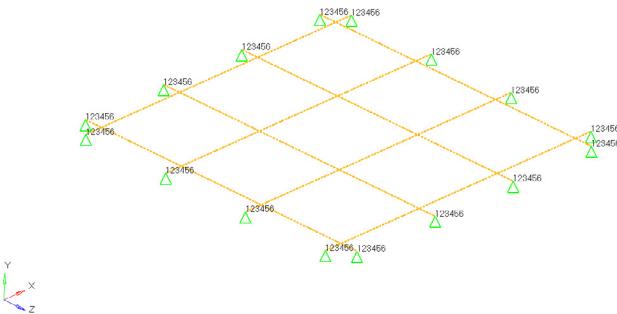
Select the elements to be updated. Make sure that rod= is set to the element type CROD,(bars= CBEAM/CBAR) and activate update.

Making use of the card edit option, , again provides the following updated element information:



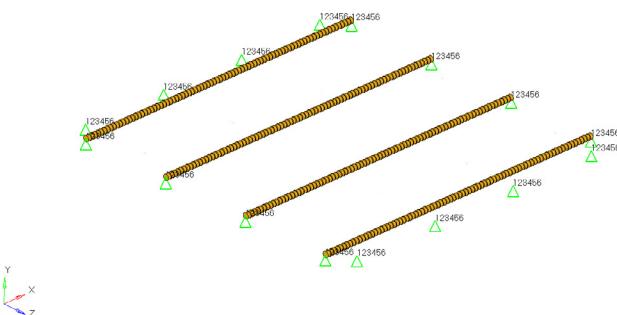
In case PID is not shown, make sure that a property collector is assigned to the CROD elements.

1D Elements Are Not Properly Displayed. What Is Wrong?



In this image, the 1D elements are displayed by using the traditional element representation . Additionally, the element shrink option, , is activated.

However, when activating the 3D element representation, , only some of the 1D elements are displayed with respect to their properties (i.e. cross-sections).



What Is The Issue With The “Missing” 1D Elements?

In the first step, make sure that a property is assigned to ALL the 1D elements. This can be checked visually. For instance, by activating the element display mode

By Prop:



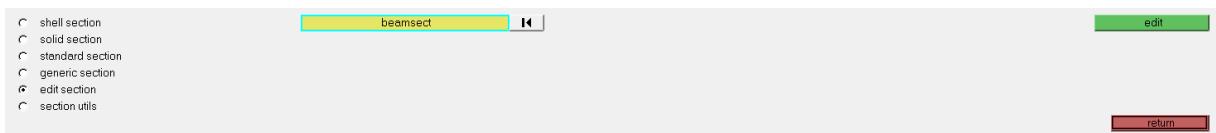
If a property collector is assigned to the 1D elements, they will be displayed with the same color as the property collector.

To better understand the properties of the 1D elements, just review the property collector (in the Model Browser make a right mouse button click on the corresponding property collector and select Card Edit):

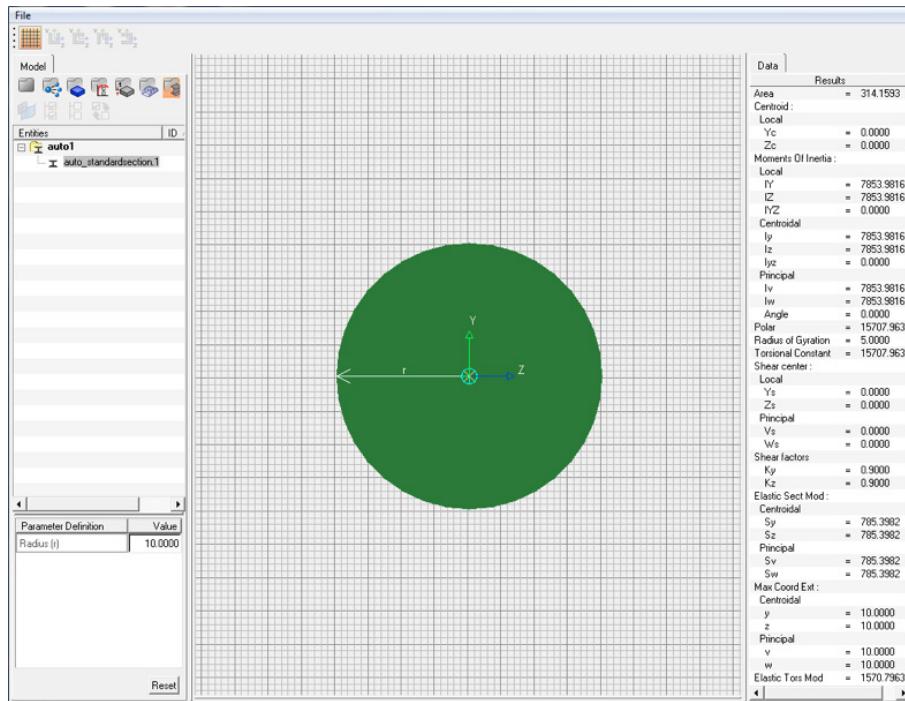
beamsec										
Pick a Beam Cross-Section =>	2									
PBEAM	PID	MD	Aa	I1a	I2a	I12a	Ja	[NSMa]		
	4	1 3 1 4 . 1 5 9 2 7 8 5 3 . 9 8 1 7 8 5 3 . 9 8 1 0 . 0 0 0 0 0 0 1 5 7 0 7 . 9 6	[M1a] [M2a] [M1b] [M2b]	N1a	N2a		[N1b] [N2b]	0 . 0 0 0 0 0 0 0 . 0 0 0 0 0		

In this particular case, the Card Image of the property collector is PBEAM (so far we have been using PBARL). Also a beamsection (with the ID = 1), and a material (ID=1) is referenced. This looks quite reasonable.

Even though all of the 1D elements have been assigned the same property (see check number 1), let's have a look at the cross-section. To do this, we open up HyperBeam (Properties > HyperBeam), select the edit section subpanel and select the beamsection of interest.



This will visualize the cross-sectional properties section (radius, area, moment of inertia etc.) as shown below.

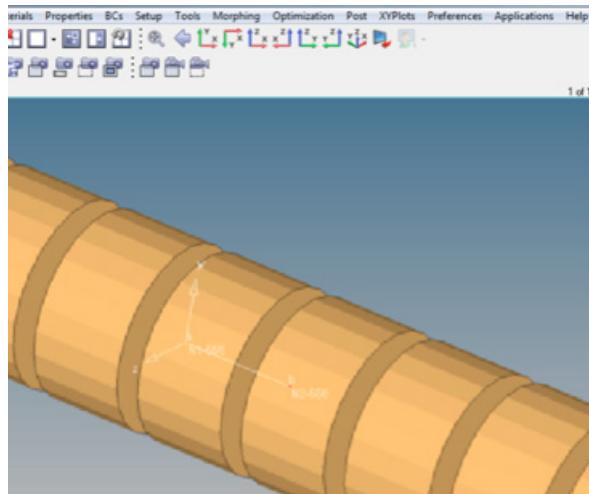


This looks reasonable too.

Recall, that the element orientation is defined through the elements local xy-plane. The local xy-plane in turn is defined via the elements x-axis (longitudinal direction from node a to b) and a vector v (more information is also available in the help documentation Index > CBEAM or CBAR).

To control (or update) the local element xy-plane and thus the v vector, go to the Bars panel (Mesh > Edit > 1D Elements > Bars)

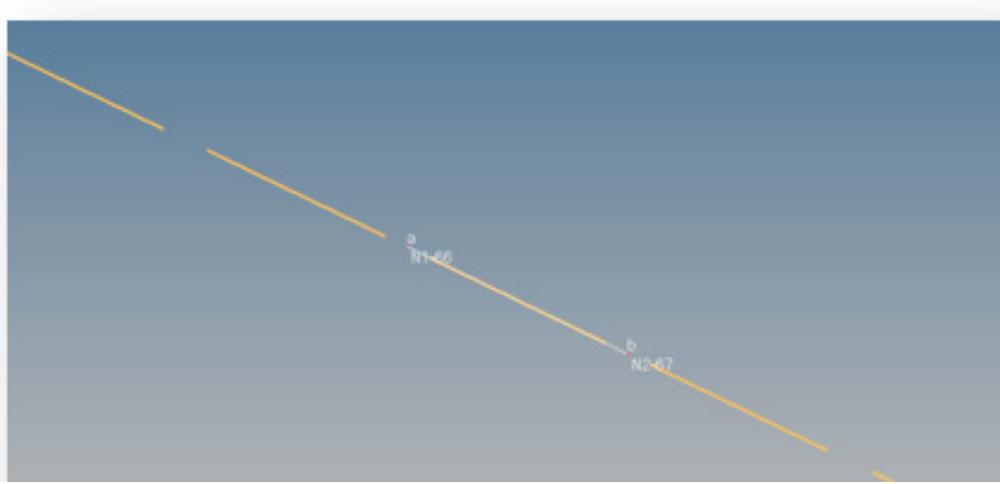
and select any of the displayed 1D elements (in the image below the 3D Element Representation  is active)



As shown above, this will display not only the local element coordinate system but also the components of the vector v (with respect to the global coordinate system). Here, the vector v parallels the global y -direction (y comp = 1). The element xy -plane thus parallels the global xy -plane.

In the next step, we are reviewing the element orientation of a 1D element which is not properly displayed (cross-sections are not shown). You may need to switch back to the standard visualization mode .

Then, just repeat the “review steps” from before.



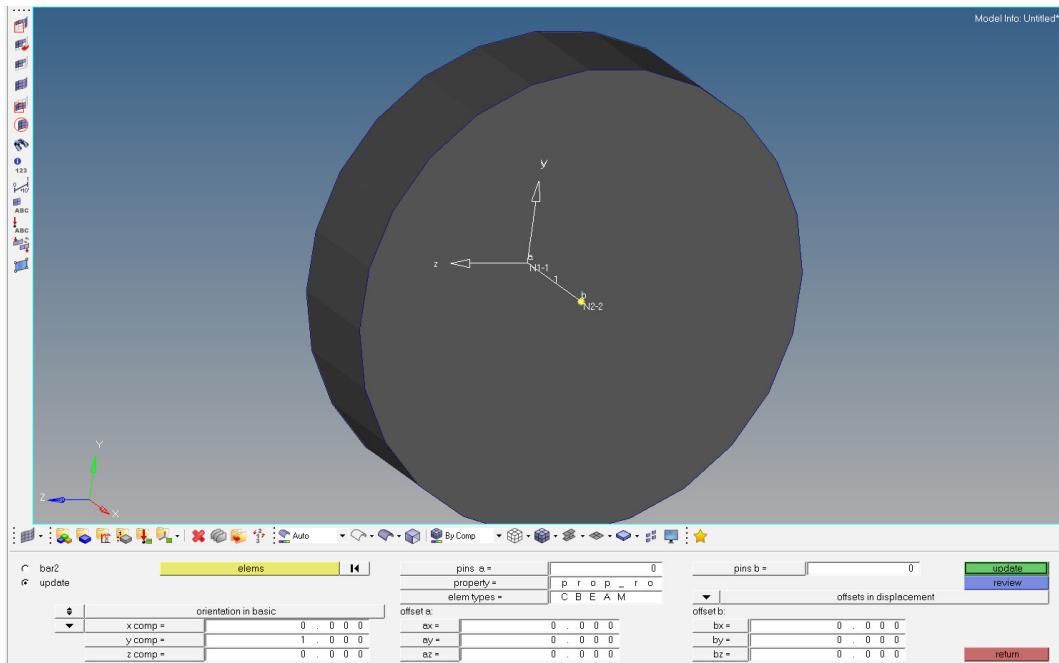
This time the location of node a and b is shown only. In the panel shown below it becomes apparent that the components of the v vector are all zero.

<input type="radio"/> bar2	<input checked="" type="radio"/> elems	<input type="button" value="update"/>
<input type="radio"/> update		
orientation in basic		
x comp =	0 . 0 0 0	
y comp =	0 . 0 0 0	
z comp =	0 . 0 0 0	
pins a =		
property =	0	
elem types =	prop - ro	
offset a:		
ax =	0 . 0 0 0	
ay =	0 . 0 0 0	
az =	0 . 0 0 0	
pins b =		
property =	0	
elem types =	prop - ro	
offsets in displacement		
bx =	0 . 0 0 0	
by =	0 . 0 0 0	
bz =	0 . 0 0 0	
<input type="button" value="update"/>	<input type="button" value="review"/>	<input type="button" value="return"/>

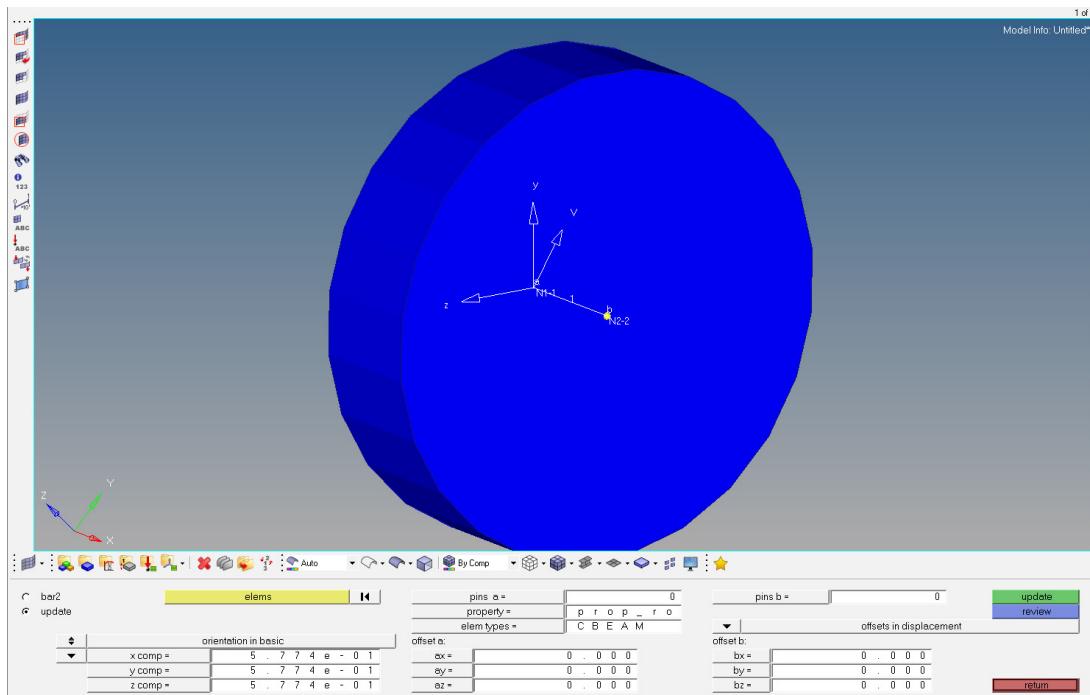
The components of vector v can be corrected/updated at any time.

Select one or all of the elements, and then specify the component of the vector v with respect to the global (basic) coordinate system. In case the y -component of vector v is set to $y=1$ (global y), both, vector v and local y -direction of the element coincide.

In the image below the local y direction and v vector are superimposed on each other.



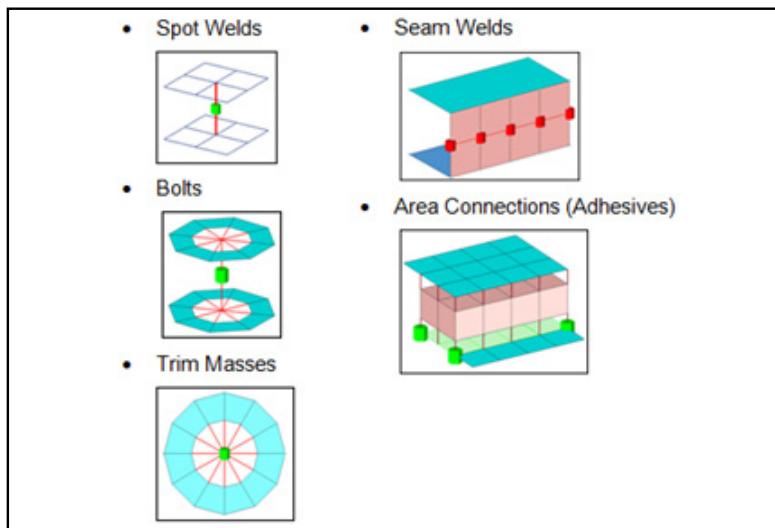
As indicated in the next image, the vector v doesn't coincide with any of the global components. Now it is quite apparent, that vector v and local x-axis together specify the local xy-plane of the element.



10.6 Connectors In HyperMesh

Connectors - What Are They?

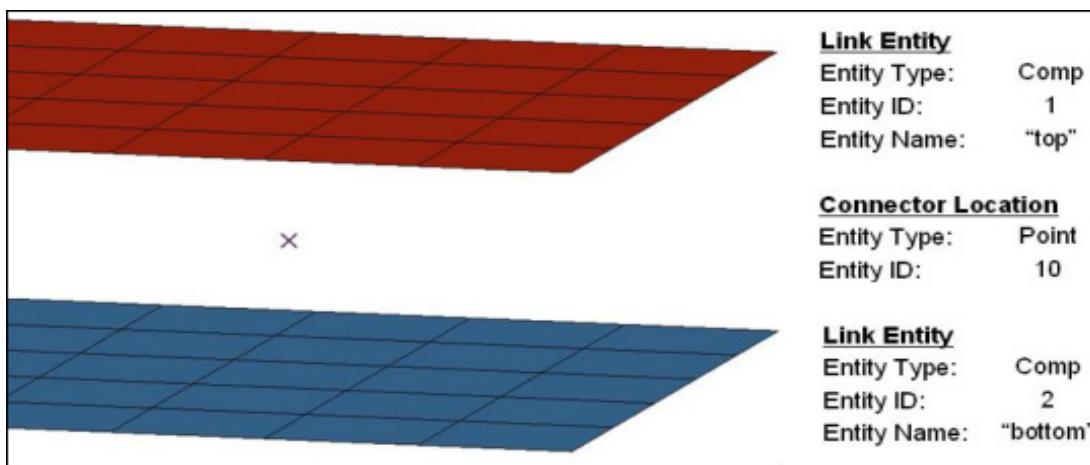
Connectors are a geometric representation of connections between entities. The advantage of connectors is the ability to create multiple connections at a single time. Hundreds or even thousands of connections that would normally have to be created manually one at a time can be mass created, even before the part is meshed. They can be used to create numerous types of connection elements such as:



Connector - Definition

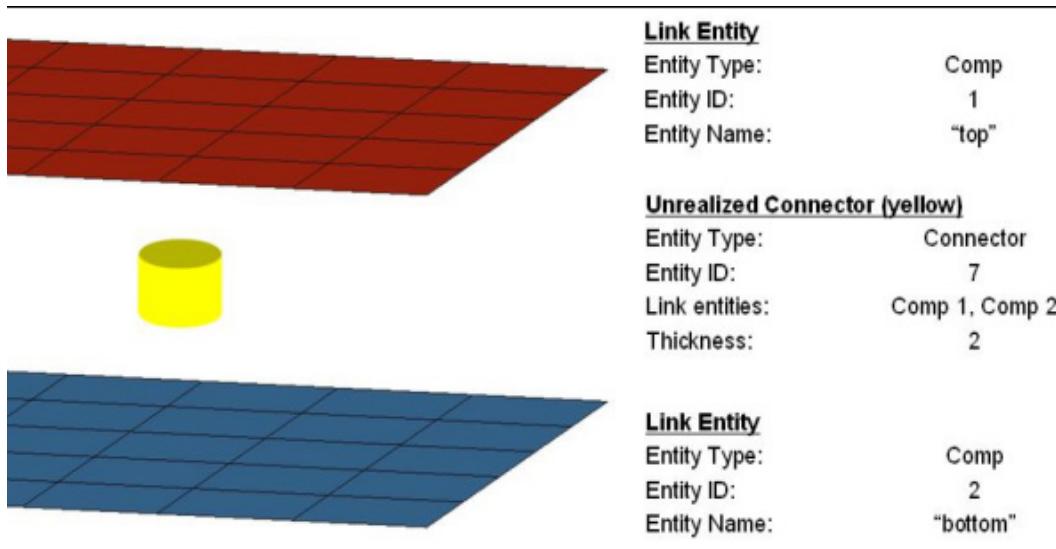
The connector is simply a database of information defining a specific request for connection at a specific location. A connector definition describes the connector between multiple entities at a specific location. Entities that are to be connected are referred to as link entities. The connector location can be defined as a node or node list, a geometric point, a line or line list, a surface, or even as elements or tags.

In the following example, there are two components (Top and Bottom) that are to be connected at the location of a point (with an id of 10). In this case, both components are considered to be link entities, since they are to be linked together. The point defines the location of the connector.



After a connector is created, the connector icon is placed at point 10, and components 1 and 2 are incorporated into the request for connection.

The following diagram shows the connector after it is created (with an id of 7) at the location of point 10 (point 10 is not visible).



In this example, connector 7 has been defined and no welds have been created. The connector stores the following information:

- Which link entities the request for connection is to connect (Comp 1 and Comp 2)
- The thickness of the realization (Thickness = 2)
- Where to connect the link entities (the connector's current location)

HyperMesh entities currently supported as link entities include ASSEMS, COMPS, ELEMS, SURFS, NODES, and TAGS. Any number of link entities of differing types can be added to a connector in any order. The connector sets the order of link entities during the realization process.

The example above is a simple case where we have added two link entities of the same type (COMPS) to a single connector. For more detailed cases of connecting assemblies, see Example of Connecting Assemblies.

Connectors Terminology

- Link Entities - The entities that are being connected
- Connector Location - Where the entities are linked
- Connector Realization – The creation of the finite element representation of that connector
- Connector State – Whether an FE representation of a connector has been created
- Unrealized - The initial status of the connector entity upon creation
- Realized - The status only if creation of the FE weld representation at the connector was successful
- Failed - The status if creating the FE weld representation at the connector was not successful
- # of Layers – Number of FE weld layers to attempt to generate for the connector
- Connect When – Specifies when the link entity information is added to the connector

10.7

Learn More About Connectors

Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

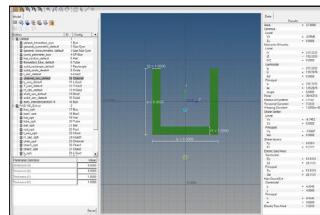
1D Elements and Connectors



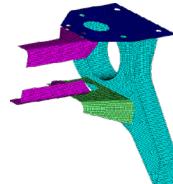
- 1D Meshing



- HyperBeam



- Connectors



1

Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

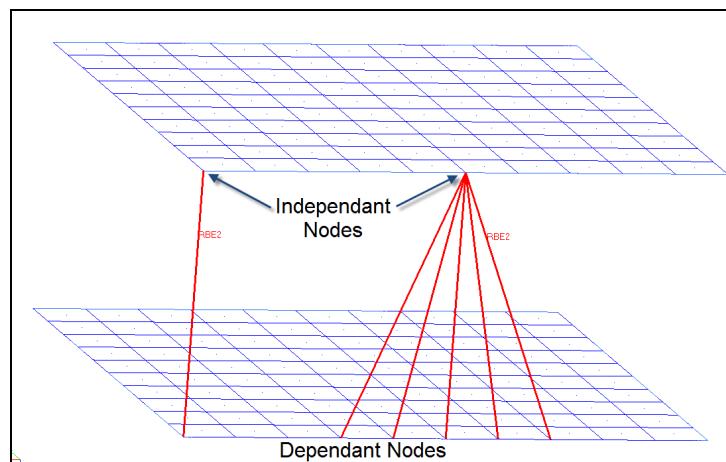
1D Elements



- 1D Elements are simple connections between nodes.
- Standard 1D Elements are

RBE2

Rigid link to transfer motion from the independent node to the dependant node(s)

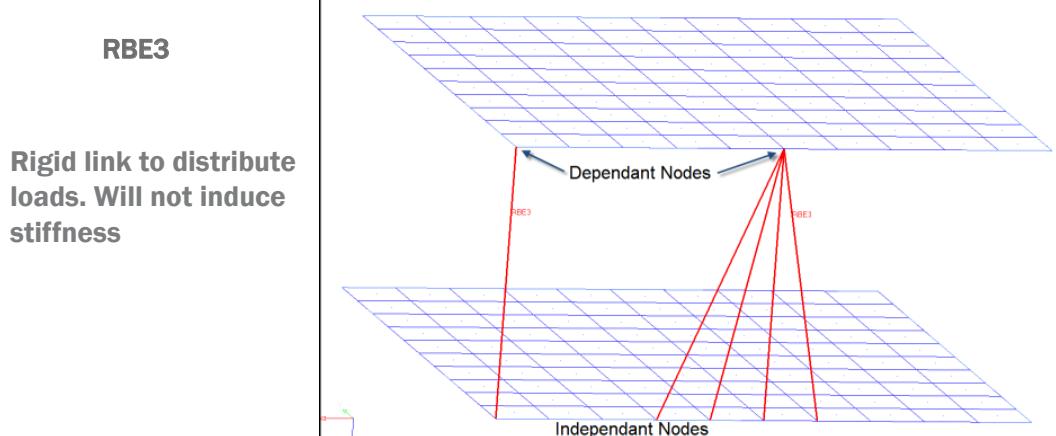


2

1D Elements



- **1D Elements** are simple connections between nodes.
- Standard **1D Elements** are

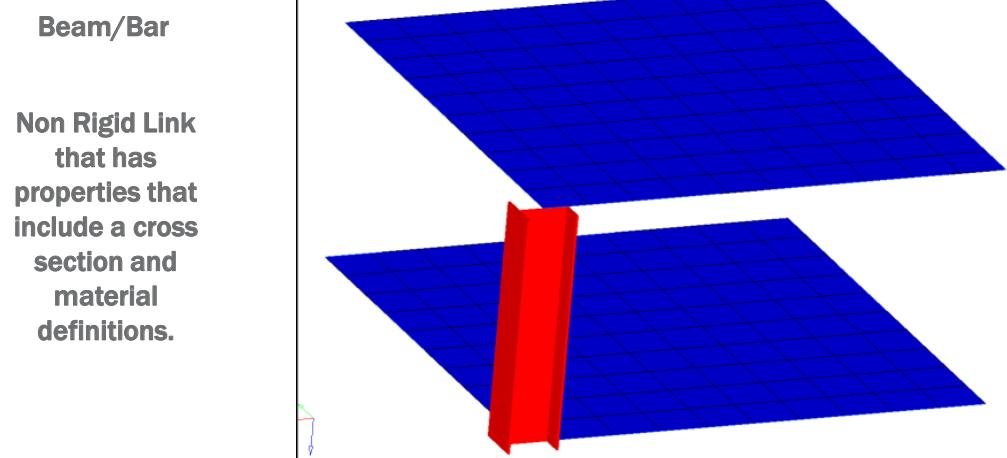


3

1D Elements



- **1D Elements** are simple connections between nodes.
- Standard **1D Elements** are



4

Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

1D Meshing – Rigid



- “Rigid” Panel → Mesh > Create > 1D Elements > Rigid

create

independent: node
dependent: node

dof1, dof2, dof3, dof4, dof5, dof6

attach dependent nodes as a set

elem types = R B E 2

update

elems

dof, connectivity, switch, attach/detach set, elem type

dof1, dof2, dof3, dof4, dof5, dof6

combine

elems

common independent node

combine, reject, review

return

5

Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

1D Meshing – Bars



- “Bars” Panel → Mesh > Create > 1D Elements > Bars

bar2

node A, node B

orientation: N1, N2, N3

pins a =
property =
element types = C B E A M

offset a:
ax = 0 . 0 0 0
ay = 0 . 0 0 0
az = 0 . 0 0 0

pins b =
offsets in displacement:
bx = 0 . 0 0 0
by = 0 . 0 0 0
bz = 0 . 0 0 0

reject, review

return

Pick a Bar Cross-Section =>		beamsec	To retrieve sections created in HBeam		Non structural mass		
PBAR	PID	MID	A	I1	I2	J	NSM
	1	0	0 . 0 0 0	0 . 0 0 0	0 . 0 0 0	0 . 0 0 0	0 . 0 0 0
	Prop ID	Mat ID	Bar section	Inertia Y	Inertia Z	Polar Inertia	

1D elems able to simulate structure with inertia, neutral axis pass through section shear center (not necessary for beams), they feel axial stress, torsion, bending.

Usual use: bolt simulation

6

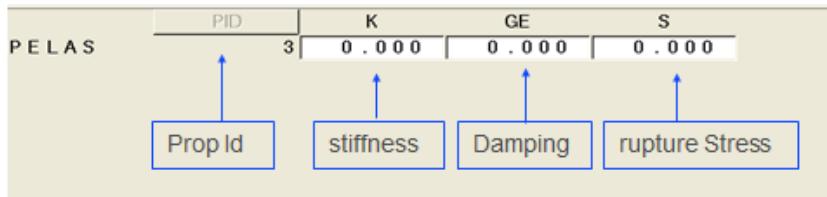
Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

1D Meshing – Springs



- “Springs” Panel → Mesh > Create > 1D Elements > Springs

<input checked="" type="radio"/> create	node node	<input type="radio"/> dof1 <input type="radio"/> dof2 <input type="radio"/> dof3	<input type="radio"/> dof4 <input type="radio"/> dof5 <input type="radio"/> dof6	reject review
<input type="radio"/> update	no vector			
		property =		
Allow to set spring direction vector		elem types =		CELAS1
return				



Elements characterized by linear stiffness

Any possible property: PELAS, PBUSH

Usual use: structures with linear elastic behavior

7

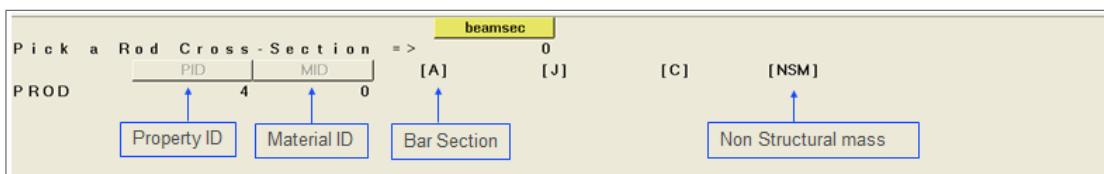
Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

1D Meshing – Rods



- “Rods” Panel → Mesh > Create > 1D Elements > Rods

<input checked="" type="radio"/> create	node node	property =	reject review
<input type="radio"/> update			
elem types = CWE LD			
return			



They have a section definition and feel axial stress and refer a material

Their property is PROD

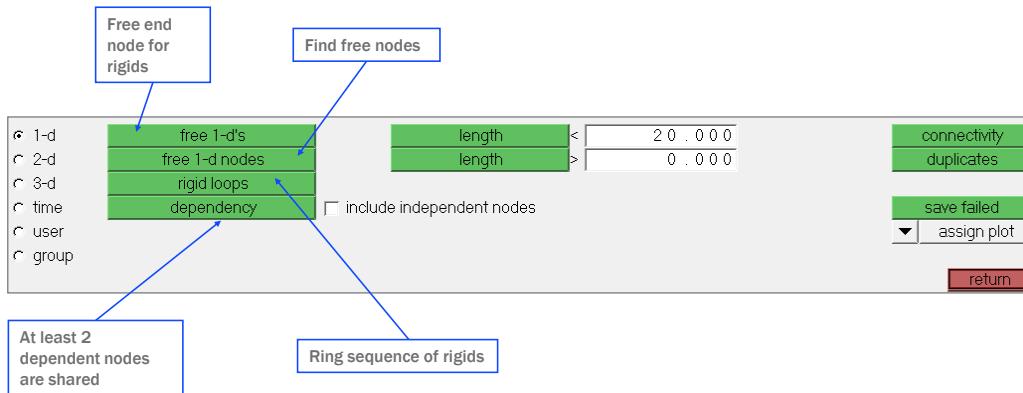
8

1D Meshing – Check Elements



- “1D Check Elements” Panel →

Mesh > Check > Elements > Check Elements > 1-d



9

1D Meshing – HyperBeam

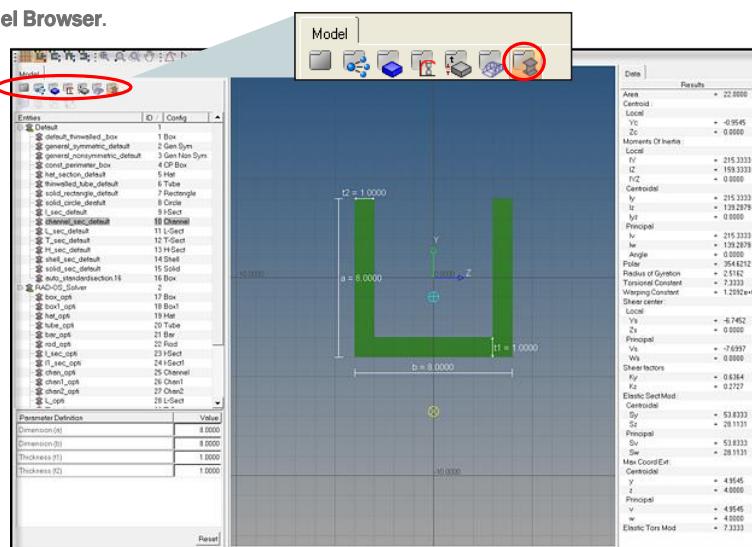


- “HyperBeam” Tool → Properties > HyperBeam

The HyperBeam panel allows you to create beam cross-section entities that you can use to simplify complex portions of your model into simple bar elements.

Most creation and editing functions are available in the **HyperBeam View** (the icon).

Model Browser.



10

Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

1D Meshing – HyperBeam



1. View in the Model Browser:

HyperBeam View

2. Create, edit, delete while in

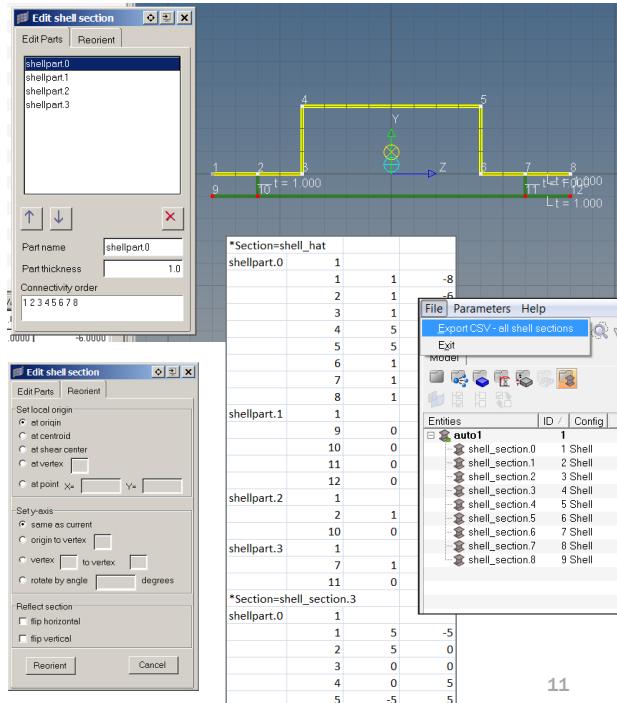
HyperBeam View

- Beamsection collectors
- Standard sections [solver libraries]
- Generic sections
- Shell and solid sections [now with the HB sketcher]

3. Shell Section Part Editor and

Orientation Editor

4. Export CSV



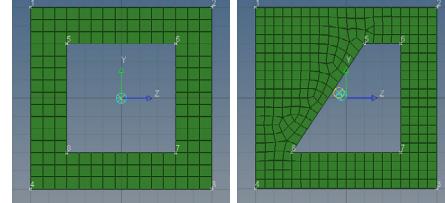
Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

1D Meshing – HyperBeam



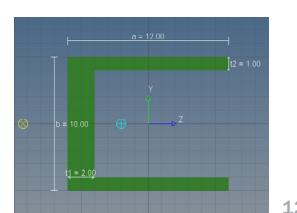
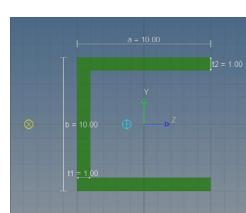
5. Section View Manipulators

- Zoom, circle zoom, pan and fit
- Hooked to the mouse in standard HM fashion



6. Vertex/parameter controller

- Add/Delete Vertex for Shell and Solid Sections
- Delete Shell Parts or Solid Perimeters
- Text Editing on screen
- Ability to snap vertices to grid locations [shell and solid sections]
- Slide thicknesses to change them
- Slide parameters for standard sections
- Auto-Weld for Shell Sections



12

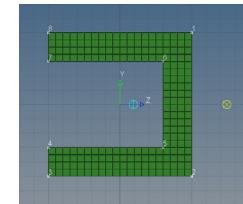
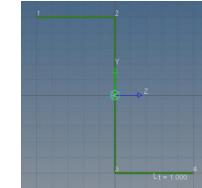
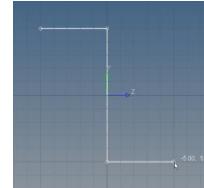
Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

1D Meshing – HyperBeam



7. Shell and Solid Section Ske

- Simple interface
- Automatic creation
- Preview Referenced and Unreferenced beamsections



Allow you to create new sections and assign them to their props:

- 1) SHELL SECTION: Section definition from lines or elements
- 2) SOLID SECTION: Section definition from lines, surfs or elements
- 2) STANDARD SECTION: Predefined sections
- 3) GENERIC SECTION: New sections managed by parameters
- 4) EDIT SECTION: Changing section properties
- 5) REVIEW UTILS: Section orientation respect to axial direction

- shell section
- solid section
- standard section
- generic section
- edit section
- section utils

save referenced
save unreferenced

return

13

Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

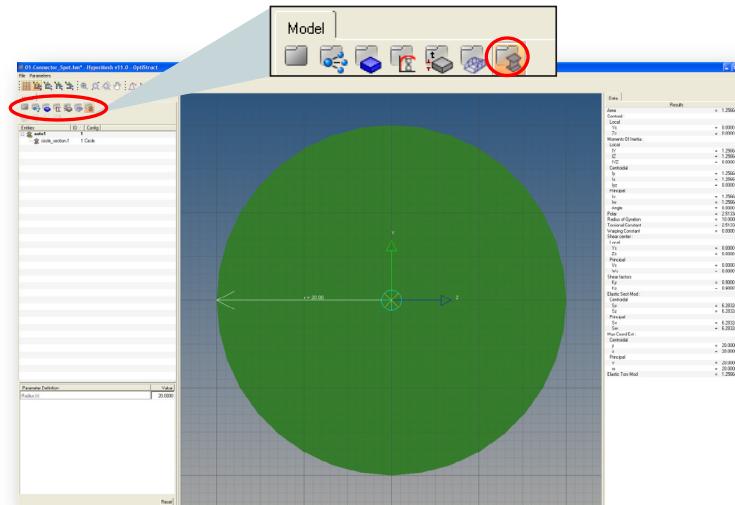
1D Meshing – HyperBeam



- “HyperBeam” Tool → Properties > HyperBeam

Example: Solid circle section to be associated to a beam

<input type="radio"/> shell section	standard section library:
<input type="radio"/> solid section	HYPER BEAM
<input checked="" type="radio"/> standard section	standard section type:
<input type="radio"/> generic section	solid circle
<input type="radio"/> edit section	
<input type="radio"/> review sections	



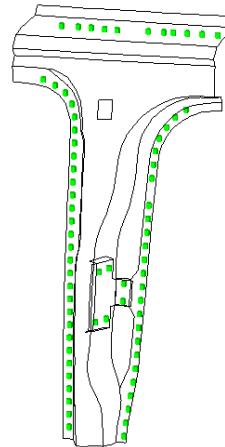
14

Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

Connectors: What are they & why use them?



- Geometric entities that define connections between other entities
 - Used for welding, bolts, adhesives, etc.
 - Link geometry or FE entities
 - Can be “realized” into FE representations of welds, etc. for any supported solver
 - rigids, CWELDs, MAT100s, ACMs, etc.
- Connectors allow for easy and rapid:
 - Creation of FE welds, bolts, adhesives, etc.
 - interactively or from:
 - XML file
 - MCF (master connectors file)
 - Spot welds only
 - Switching between different FE configurations of welds, bolts, adhesives, etc.
 - Part swapping and reconnection
 - Replacement by names or IDs



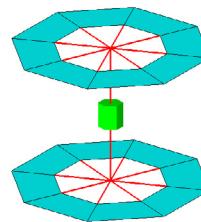
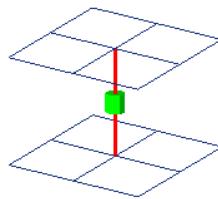
15

Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

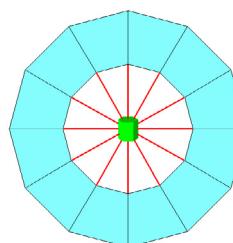
Connectors - Types of Connectors



- Spot
 - Connections at a point
 - Spot welds, rivets, etc.
- Bolt
 - Connections at hole locations
 - Bolts



- Trim Mass
 - Non-physical mass
 - Simplification through replacement of parts with mass only or add mass to existing part.



16

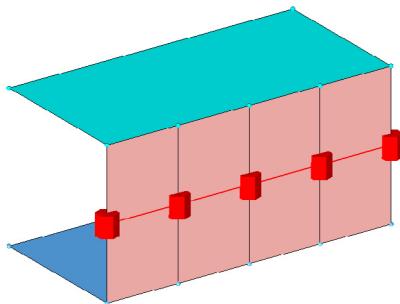
Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

Connectors - Types of Connectors



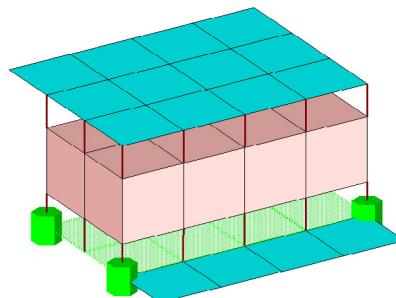
- **Seam**

- Connections along a length
- Seam welds
- Connects geometry only



- **Area**

- Connection of areas
- Adhesives



17

Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

Connectors - Terminology



- **Link Entities** - The entities that are being connected

- User can explicitly define link entities or specify a search tolerance
- Can be components, elements, surfaces, nodes, or tags
- Typically components are linked

- **Connector Location** - Where the entities are linked

- Nodes – created at the node location
- Points – created at the point location
- Lines – created for the line
 - line may be split into multiple projection locations as specified by the offset, spacing, and density values
- Elements – created at the element location (adhesives only)
- Surface – created at the surface location (adhesives only)

- **Connector Realization** – The creation of the finite element representation of that connector

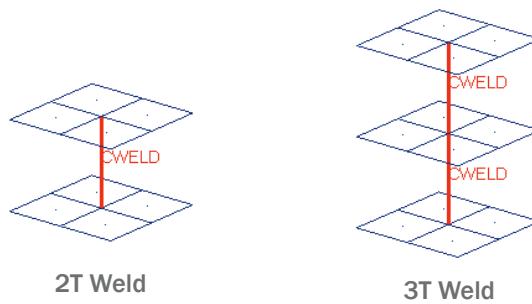
- Rigid, springs, etc., or custom configurations such as ACMs, CWELDS, etc.

18

Connectors - Terminology



- **Connector State** – Whether an FE representation of a connector has been created
 - *Unrealized* - The initial status of the connector entity upon creation
 - *Realized* - The status only if creation of the FE weld representation at the connector was successful
 - *Failed* – The status if the creating the FE weld representation at the connector was not successful
- **# of Layers** – number of FE weld layers to attempt to generate for the connector
 - 2T, 3T, etc.



19

Connectors - Terminology



- **Connect When** – Specifies when the link entity information is added to the connector
 - *Now* - Allows you to add link entity information now. For this option, you must select the *connect what* entities and *num layers* to successfully create a connector.
 - *At FE Realize* - The link entities to the connector are determined while realizing the connector. The link entities are determined by the projections and proximity from the connector location.
- **Re-Connect Rule** – Defines method for connector re-attachment during part swapping/replacement
 - *None* - If a link entity is deleted, the link entity is removed from the connector
 - *By ID* - If a link entity is deleted, the connector retains the ID of the link entity, and will map to a new entity with that ID upon realization
 - *By Name* – Same as the by id rule except that the entity name is retained

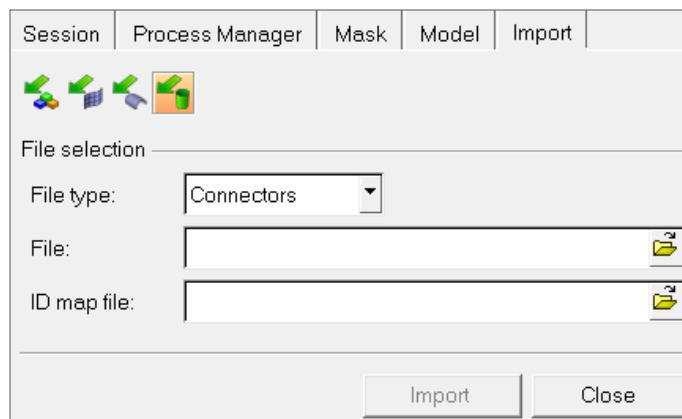
20

Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

Connectors - Tools



- File > Import > Connectors
 - Import Master Weld Files to automatically create connectors



21

Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

Connectors - Tools



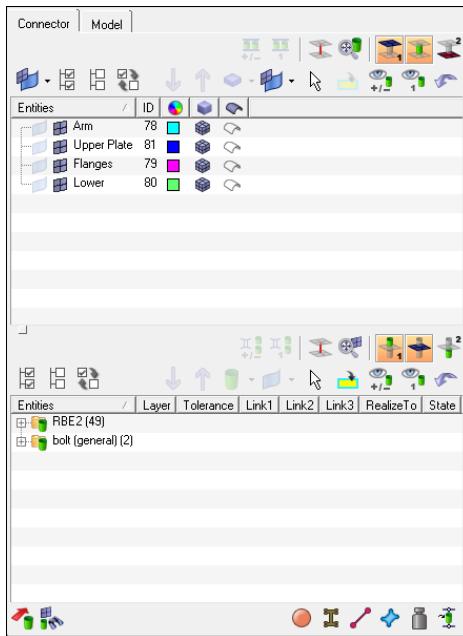
- Connectors Tools (Connectors > Create >)
 - Spot } create, edit, and realize that style of connectors
 - Bolt
 - Seam
 - Area
 - Apply Mass – adds a mass value to entities
 - Used to represent mass of parts that are not present in the model
 - FE Absorb – Create new connectors from existing elements of recognizable FE representations of welds, bolts, adhesives, etc.
 - Add Links – Add link entities to existing connectors
 - Unrealize – Delete FE representations of welds / bolts / adhesives associated with existing connectors
 - Compare – Checks the MCF against displayed model file
 - Quality – Check for duplicate connectors, combine connectors, check quality of realized elements

spot	add links	compare	connector options
bolt	unrealize	quality	
seam	mesh edit		
area			
apply mass			
fe absorb			

22

Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

Connectors - Tools



- **Connector Browser** – Hierarchical browser that provides information and the ability to edit connectors
 - Shows:
 - Type of Connector
 - Link Information
 - State of connector
 - Editable
 - Edit link entities, export mwf files, etc
 - Found in Tab Browser Area

23

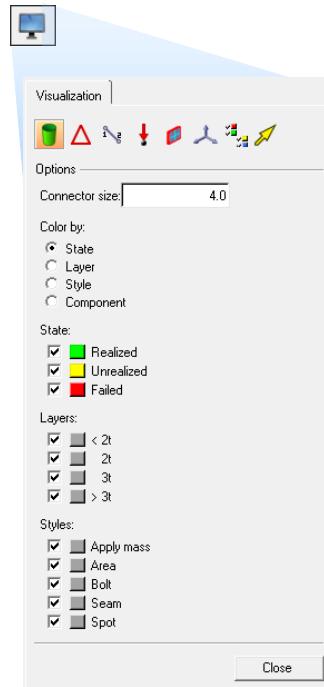
Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

Connectors Module - Visualization options



- **Visualization Options** – Controls how connectors are displayed:

- Color connectors by state, layers, or component
- Visibility by state or layers (can turn the display on or off)
- Control size of connector display



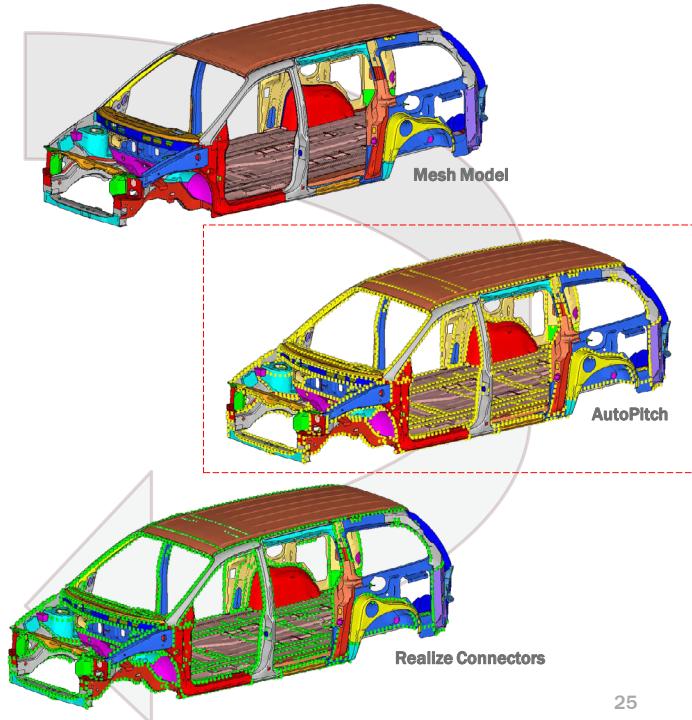
24

Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

Connectors - Autopitch



- During the model build process weld data from CAD is often unavailable.
- The Autopitch tool creates unrealized connectors at a user defined pitch distance.
- The connectors can be realized using the standard connector functionality.



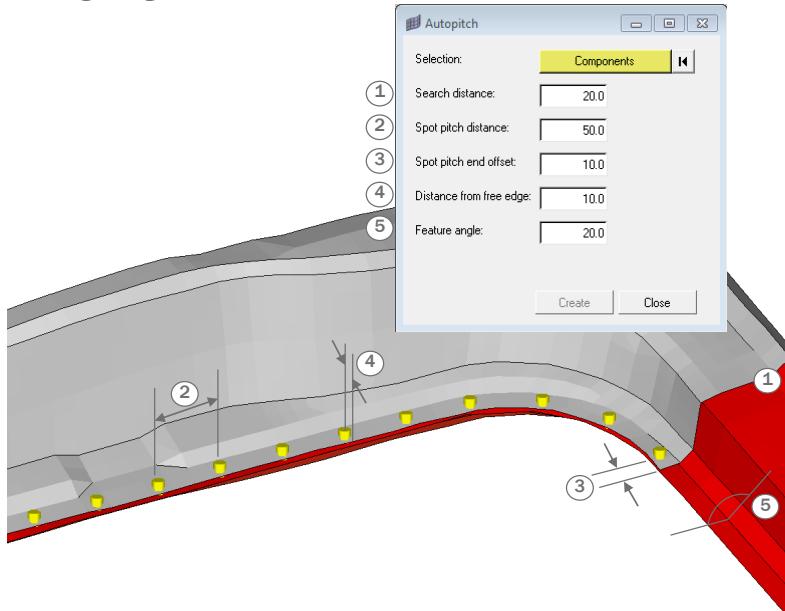
25

Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

Connectors - Autopitch



- “Autopitch” Tool → Connectors > Create > Autopitch
- Autopitch will connect flange>flange, flat>flange, and flat>flat regions – it’s more than just connecting flanges...



26

Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

Connectors Mesh Edit - Imprint



- Connectors “**Mesh Edit**” → **1D page > connectors > mesh edit > imprint**
 - When creating mesh-dependent realizations with quad transitions, the quad transition meshes can overlap and disturb each other if more than one set of connectors is created too close to each other. The **connector imprint** panel provides a way to reconcile such transitions with each other and then modify the underlying mesh to match the results. This creates a seamless, properly meshed final result.
 - The **connector imprint** panel is accessible from within the **connectors** module, which is accessible in the panel area of the **1D**, **2D**, and **3D** menu pages. It uses the standard **mesh edit** panel’s **imprint** subpanel, but defaults to different values for several of the entry fields.
 - Use of the mesh edit - connector imprint panel is generally only required when sets of connectors with quad transitions are located too close to each other for the specified mesh size

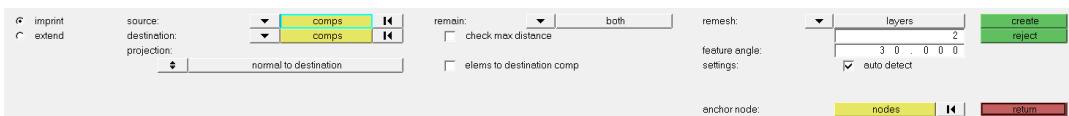
27

Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

Connectors Mesh Edit - Imprint



- Connectors “**Mesh Edit**” → **1D page > connectors > mesh edit > imprint**
 - The panel includes collectors for specifying the imprint mesh (**source**) and the **destination** mesh, as well as an option to determine which components the imprint will be organized into. There are additional settings to be made; for further details see the **mesh edit** panel.
 - The below screenshot displays the recommended panel settings for **mesh imprint** in combination with **connectors**:



28

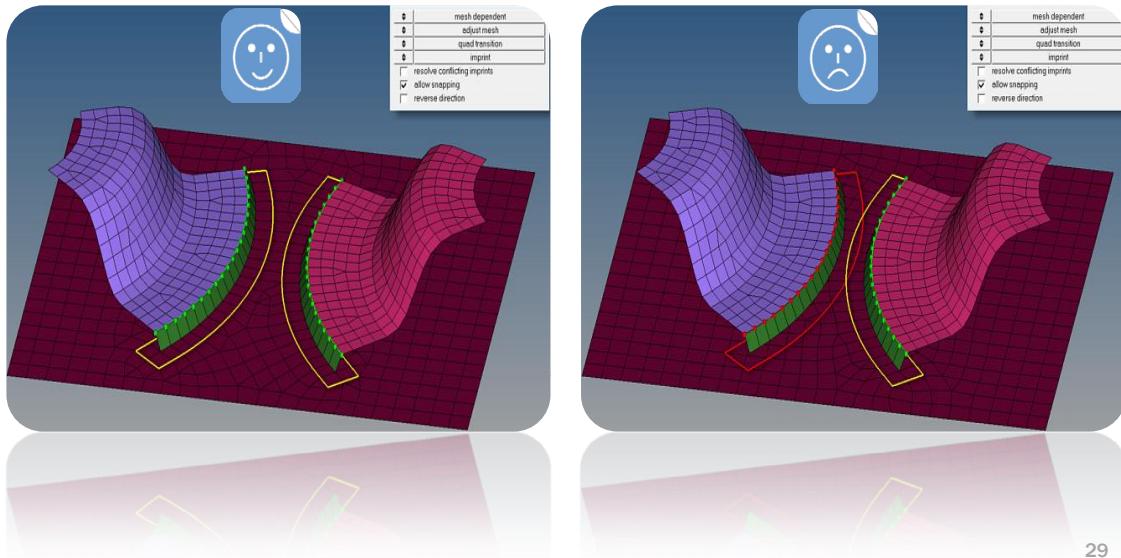
Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

Connectors Mesh Edit - Imprint



- Connectors “Mesh Edit” → 1D page > connectors > mesh edit > *imprint*

Resolve conflicting Imprint: 1° Strategy



29

Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

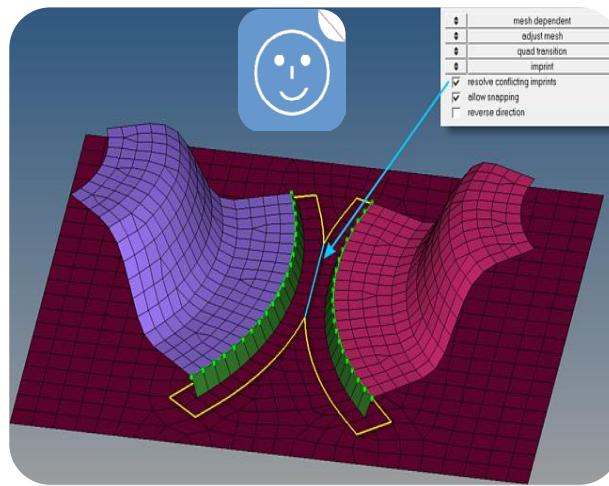
Connectors Mesh Edit - Imprint



- Connectors “Mesh Edit” → 1D page > connectors > mesh edit > *imprint*

Resolve conflicting imprints: 1° Strategy

- Smaller conflicts can be resolved automatically when realizing the connectors. This releases the overlapping elements and performs a normal remesh in that area. This is permitted as long as the overlapping area is smaller than half the regular quad transition element. However, if a conflict is too great it cannot be resolved in this fashion.



30

Copyright © 2013 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

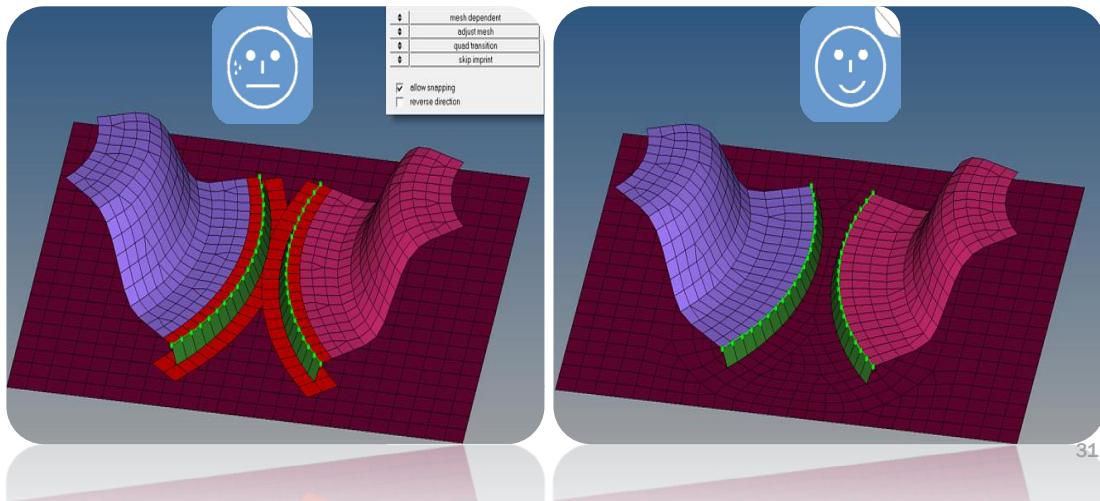
Connectors Mesh Edit - Imprint



- Connectors “*Mesh Edit*” → 1D page > connectors > mesh edit > *imprint*

Manual conflicting imprints: 2° Strategy

- In the second step, the conflicting elements can be manually modified with all of the functions that HyperMesh provides.
- Then the manual mesh imprint is performed.



31

10.8 1D Meshing Tutorials And Videos

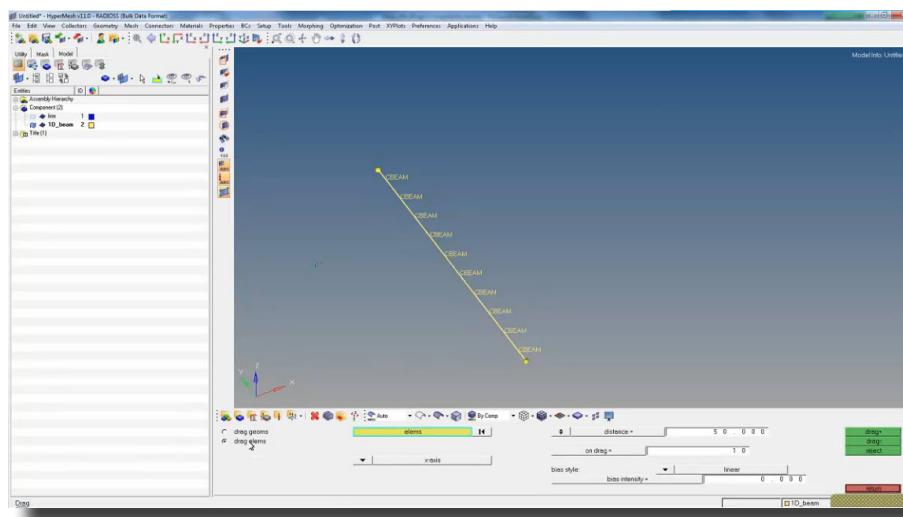
Recommended Tutorials

These tutorials are included in your HyperWorks installation.

- HM-3000: Creating 1D Elements
- HM-3400: Creating Connectors
- HM-3410: Creating Area Connectors
- HM-3420: Creating Bolt Connectors
- HM-3430: Part Replacement Through Connectors
- HM-4020: Obtaining and Assigning Beam Cross-Section Properties using HyperBeam

Recommended Webinars

Creating, checking, and visualizing 1D elements in HyperMesh



We recorded a webinar about how to use RBE and MPC elements. Likely errors and their impact on modeling results are discussed. In addition, tips are given on how to avoid such errors (<https://altair-2.wistia.com/medias/eu86k2rd0g>)

The thumbnail shows the title 'Using RBE's and MPC's in RADIOSS' in orange at the top. Below it is a small image of Kristian Holm. At the bottom, there is contact information: 'Kristian Holm', 'Product Specialist Implicit Analysis', and 'Holm@altair.de'.

Videos About 1D Elements (not only related to HyperWorks)

1D Bar element, part 1 (Prof. J. Chessa, University Texas; <http://youtu.be/zYPrPntBUEg>)

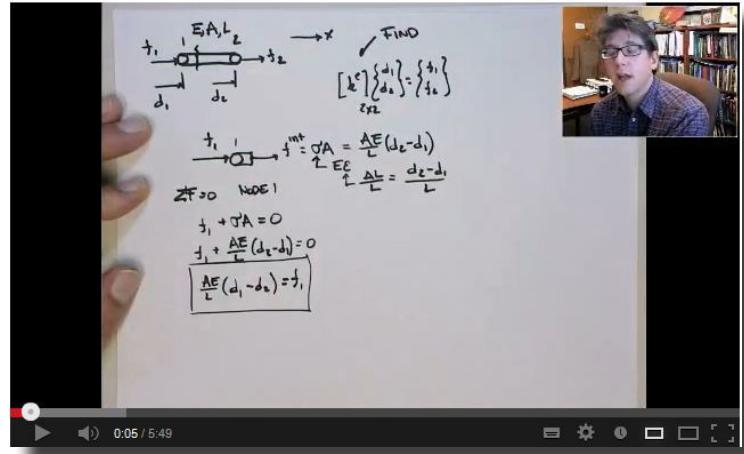
MECH 4326 Applied Finite Element Analysis

Derivation of the 1D bar element stiffness matrix
(part 1 of 2)

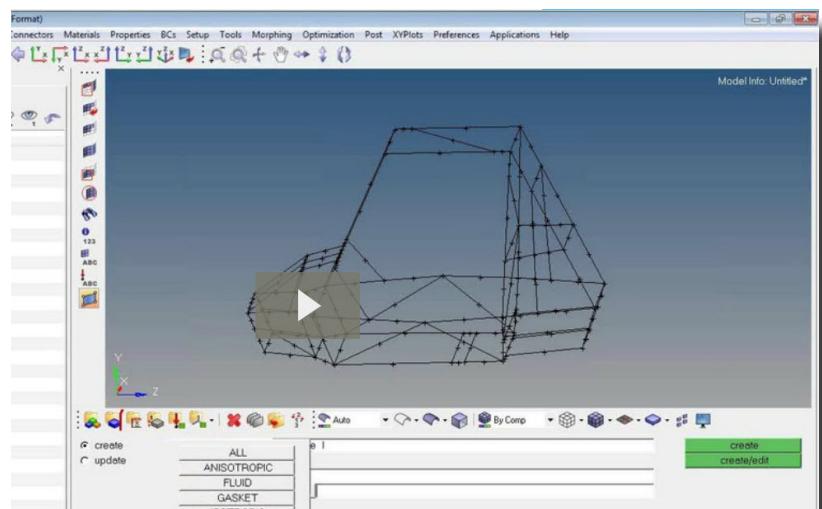
Jack Chessa (jfchessa@utep.edu)
Associate Professor
Mechanical Engineering
The University of Texas at El Paso



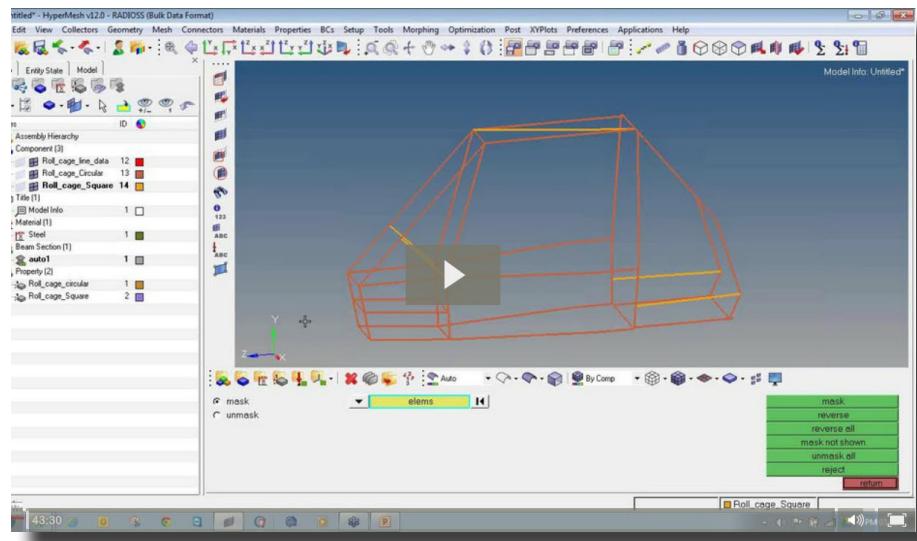
1D Bar element, part 2 (Prof. J. Chessa, University Texas; <http://youtu.be/A6R3UIe5xQY>)



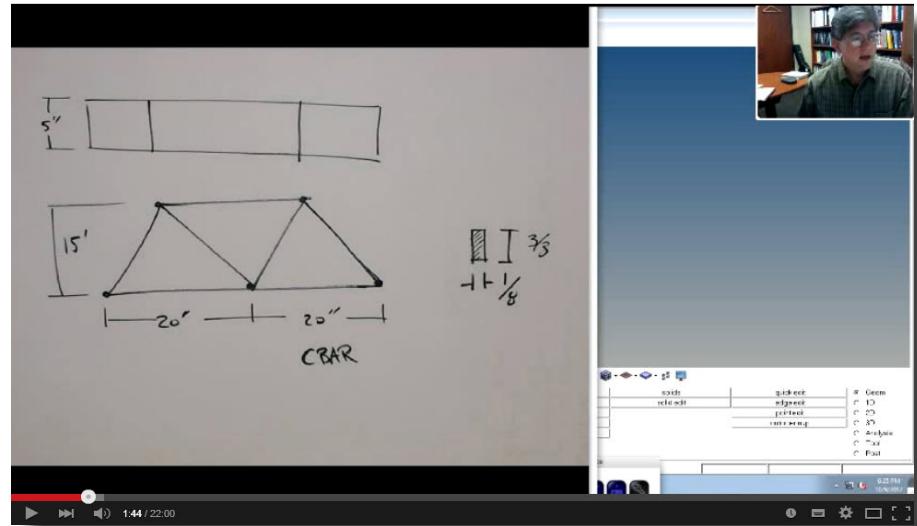
How to generate 1D Meshing for Roll Cage structure using Altair HyperMesh (by Apoorv Bapat, Altair; <https://altair-2.wistia.com/medias/gld9wndyoO>)



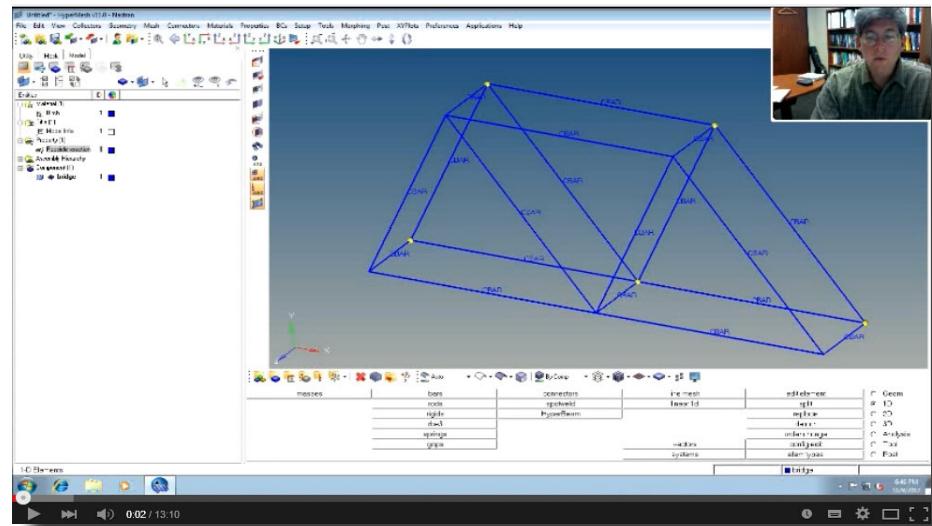
1D Meshing of a Roll Cage (advanced topics, by Apoorv Bapat, <https://altair-2.wistia.com/medias/xxihgpt4qx>)



HyperMesh for CBAR models part 1 (by Prof J. Chessa, Texas) <https://www.youtube.com/watch?v=hGmPlz1FRgo>

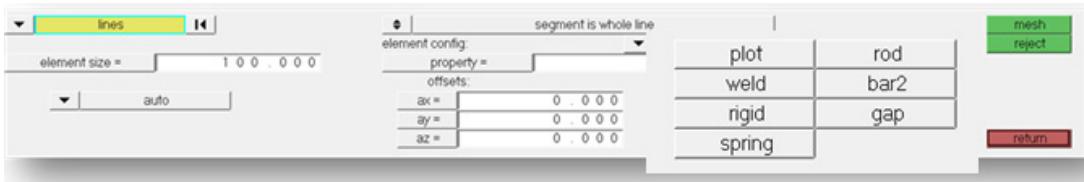


HyperMesh for CBAR models part 2 (by Prof J. Chessa, Texas; <https://www.youtube.com/watch?v=De1Z1GV6sQg>)



10.9 Student Racing Car Project - 1D Meshing

As the given geometry consists of lines, the Line Mesh panel is used for meshing. To access the Line Mesh panel, select Mesh > Create > Line Mesh from the menu bar.



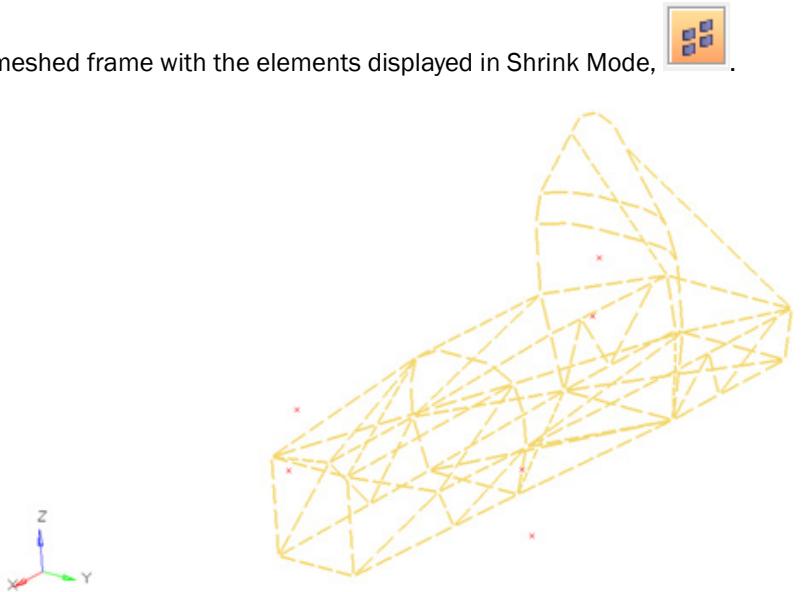
At this stage, there is no property collector available. Thus, the property field remains blank. The mean element size is set to be 100 mm and the element configuration is bar2. Bar2 elements are 1D (1st order) elements with 2 nodes used to model axial, bending, and torsion behavior. We just have to specify the kind of 1D element which should be used for the analysis.

The Element Type panel is used to set the type of 1D elements to be used. This panel is accessed by selecting Mesh > Assign > Element Type.

The element type “bars” may be set to CBEAM (default). As we are going to use CBAR elements it may be necessary to switch the element type in here.

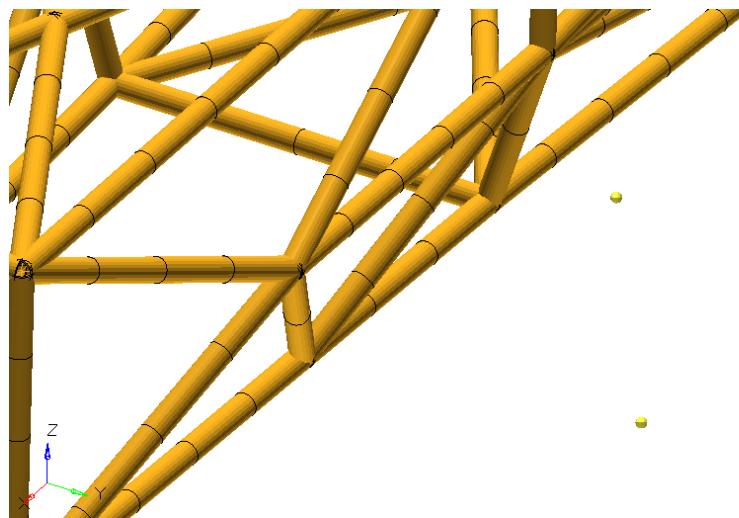
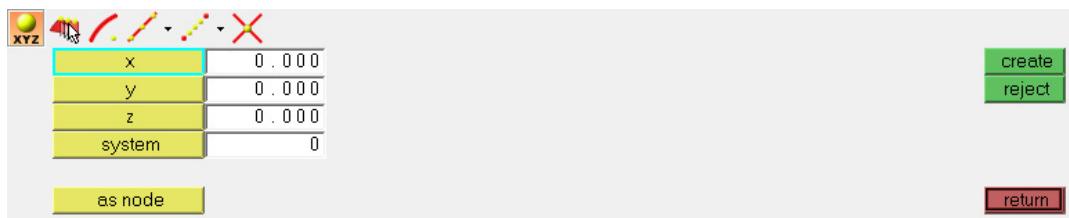


Below: image of the meshed frame with the elements displayed in Shrink Mode,

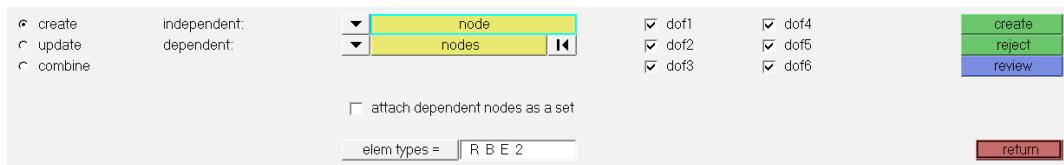


Wheel Suspension

The wheel suspension is modeled through rigid links (RBE2). However, before creating the rigid links temporary nodes at the location of the independent nodes must be created. The location of these nodes is given through the geometry points (stored in the component “points_wheel”). In the first step temporary nodes are placed at the location of the geometry points by selecting Mesh > Create > Nodes.



Temporary nodes (yellow) at the location of the geometry points. These temporary nodes are then used to create the rigid elements representing the wheel suspension by selecting Mesh > Create > 1D Elements > Rigid.



Note that the selector next to “dependent” is set to “nodes” i.e. multiple nodes.

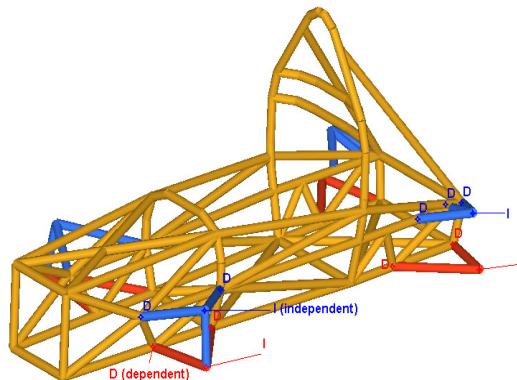


Figure: Wheel suspension modeled by means of rigid links (RBE2) with independent and dependent nodes.



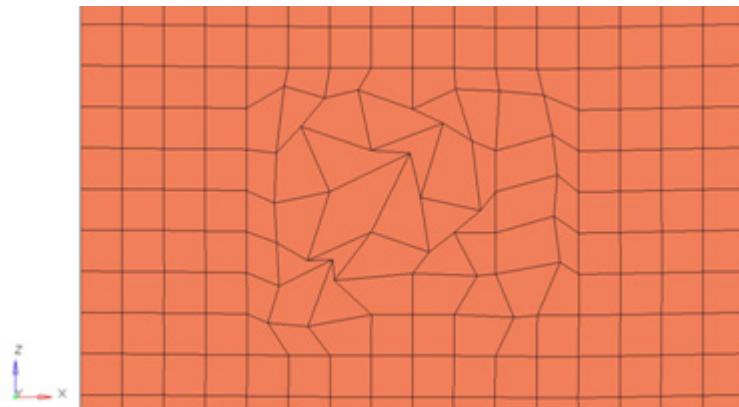
11 2D Meshing With HyperMesh

This chapter includes material from the book “**Practical Finite Element Analysis**”. It also has been reviewed and has additional material added by Matthias Goelke.

Once geometry cleanup is completed (e.g. surfaces are stitched together — no unwanted free surface edges inside the geometry), meshing is next.

Some rules of thumb when meshing:

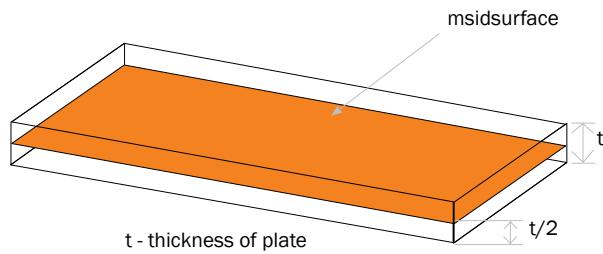
- The mesh should look rather smooth and regular (keep in mind that the analysis is based on your mesh and the mesh quality is key).
- Use the simplest element type suited for the problem.
- Start with a coarse mesh and understand the modeling results; then use a finer mesh if needed.
- Try to keep mesh related uncertainties to a minimum if possible. Keep it simple as it can get more complicated on its own.



The image above illustrates an “ugly” mesh. This chapter focuses on when to use 2D elements, how to create 2D elements of good quality, and how to use HyperMesh to create 2D elements.

11.1 When To Use 2D Elements

2D elements are used when two of the dimensions are very large in comparison to the third dimension.



Element shape: Quad, tria

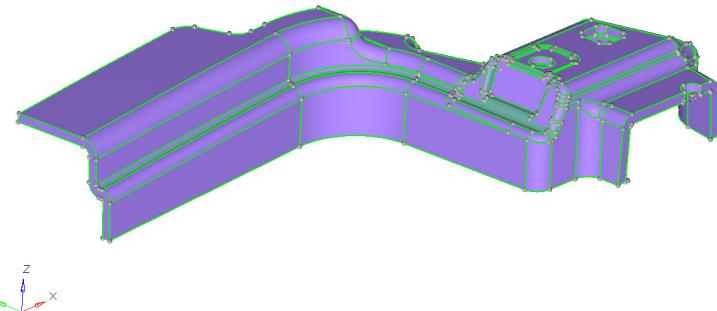
Additional data from user: Remaining dimension i.e. thickness

Element type: Thin shell, plate, membrane, plane stress, plane strain, axi-symmetric solid etc.

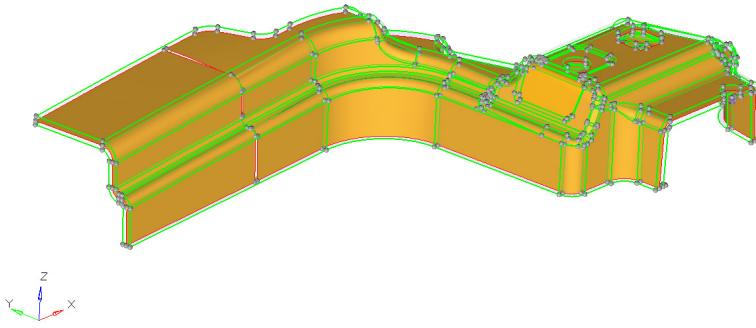
Practical applications: Sheet metal parts, plastic components like instrument panel etc.

Why Is 2D Meshing Carried Out On The Mid Surface?

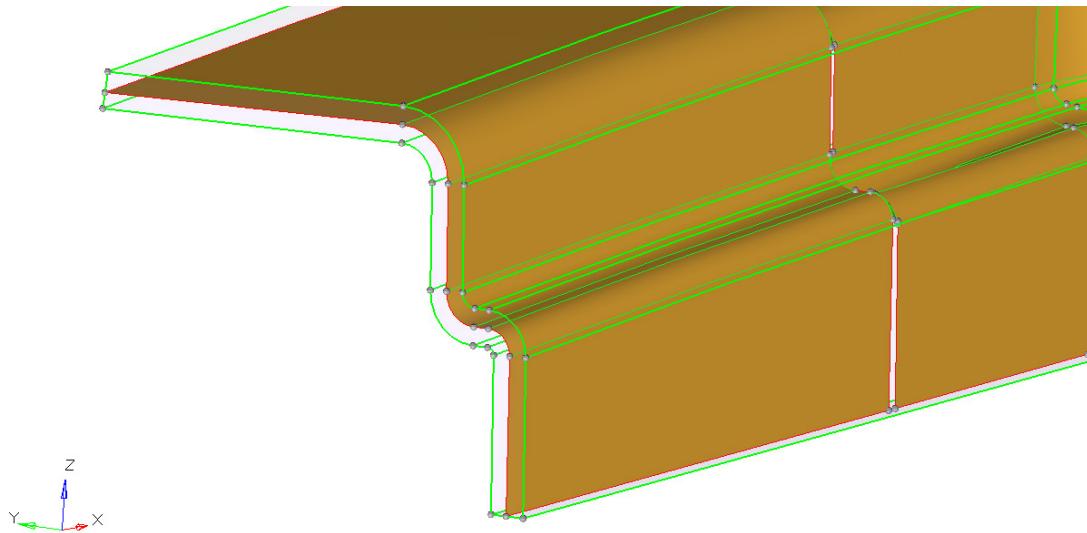
Quite often the geometry of thin walled 3D structures, as shown in the image below, is simplified to a geometric model with lower dimensionality. This is typically called a mid-surface model. The mid-surface model is then meshed with 2D elements. Thus, there is no need for a detailed volume mesh as the thickness of the geometry is virtually assigned to the 2D elements. Mathematically, the element thickness (specified by the user) is assigned with half in the + Z direction (element top) and the other half in the - Z direction (element bottom).



Given 3D thin walled geometry

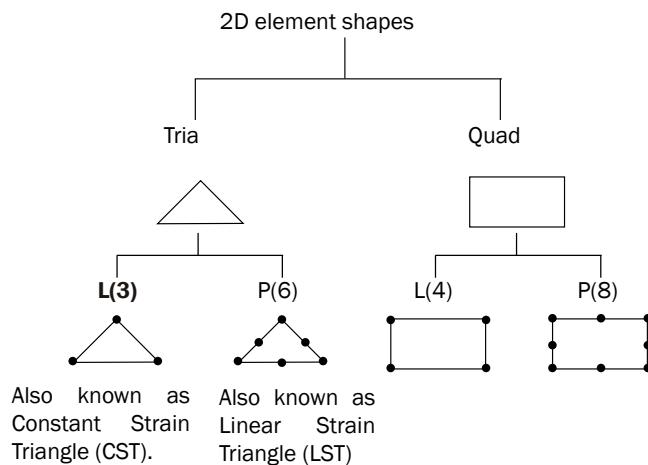


Derived mid-surface geometry



Enlarged view on the mid-surface model

Overview on 2D element shapes



* L – Linear element * P – Parabolic element

*() – Indicates number of nodes/element

Constant Strain Triangle (CST) Information

Some remarks regarding the Constant Strain Triangle (CST) element.

The explanation below is taken from:

The CST (Constant Strain Triangle) –An insidious survivor from the infancy of FEA, by R.P. Prukl, MFT (this paper is also uploaded to the Academic Blog; http://www.altairuniversity.com/wp-content/uploads/2013/06/The_CST.pdf)

The CST was the first element that was developed for finite element analysis (FEA) and 40-50 years ago it served its purpose well. In the meantime, more accurate elements have been created and these should be used to replace the CST.

The Explanation

Consider a 3-noded plane stress element in the xy-plane with node points 1, 2 and

3. The x-deflections are u_1, u_2, u_3 and the y-deflections v_1, v_2 and v_3 , totalling six values altogether.

The displacement function then has the following form (using six constants α_i to describe the behavior of the element):

$$u = \alpha_1 + \alpha_2 x + \alpha_3 y$$

$$v = \alpha_4 + \alpha_5 x + \alpha_6 y$$

The direct strains can then be calculated by differentiat $\varepsilon_x = \frac{\partial u}{\partial x} = \alpha_2$

$$\varepsilon_y = \frac{\partial v}{\partial y} = \alpha_5$$

What Does This Mean?

The strains in such an element are constants. We know, however, that in a beam we have compression at the top and tension at the bottom. Our single element is, therefore, not capable of modelling bending behavior of a beam, it cannot model anything at all. Note: Three-noded elements for other applications than plane stress and strain are quite acceptable, e.g., plate bending and heat transfer.

The Remedy

Use elements with four nodes. We then have eight constants to describe the behavior of the element:

$$u = \alpha_1 + \alpha_2 x + \alpha_3 y + \alpha_4 xy$$

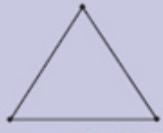
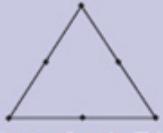
$$v = \alpha_5 + \alpha_6 x + \alpha_7 y + \alpha_8 xy$$

The direct strains are then as follows:

$$\epsilon_x = \frac{\partial u}{\partial x} = \alpha_2 + \alpha_4 y$$

$$\epsilon_y = \frac{\partial v}{\partial y} = \alpha_7 + \alpha_8 x$$

The strain in the x-direction is now a linear function of its y-value. This is much better than for the triangular element. Use triangular elements which also include the rotational degree of freedom about the z-axis normal to the xy-plane.

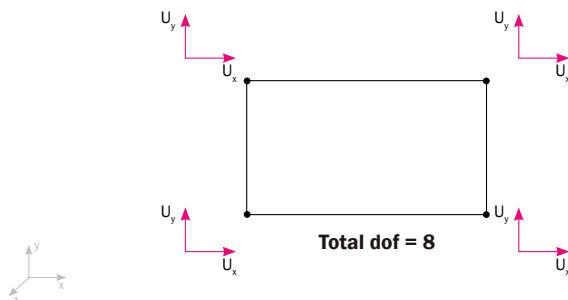
Type of Element	Displacement function
Linear Tria 3  CST (Constant Strain Triangle)	$u = a_0 + a_1 x + a_2 y$ (3 nodes – 3 terms in displacement function) $\epsilon_x = \frac{\partial u}{\partial x} = a_1 = \text{const.}$ $\epsilon_y = \frac{\partial u}{\partial y} = a_2 = \text{const.}$
Linear Quad 4 	$u = a_0 + a_1 x + a_2 y + a_3 xy$ (one additional term in comparison to tria 3, makes it more accurate)
Parabolic Tria 6  LST (Linear Strain Triangle)	$u = a_0 + a_1 x + a_2 y + a_3 x^2 + a_4 y^2 + a_5 xy$ (6 nodes – 6 terms in displacement function)
Parabolic Quad 8 	$u = a_0 + a_1 x + a_2 y + a_3 xy + a_4 x^2 + a_5 y^2 + a_6 x^2 y + a_7 xy^2$ (two additional terms in comparison to tria 6, makes it more accurate)

11.2 Family Of 2D Elements

1) Plane Stress

Degrees of Freedom (DOFs) – 2 / node $\{U_x, U_y$ (in-plane translations)}

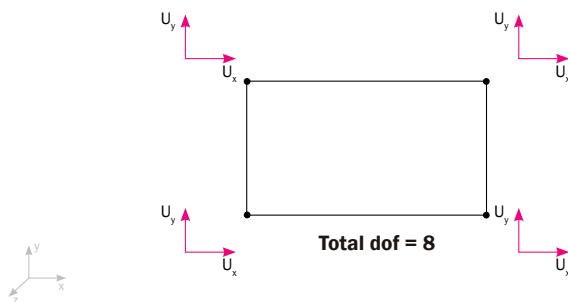
Stress in z direction (thickness) is zero ($\sigma_z = 0$)



Practical Applications: Thin sheet metal parts, like aircraft skin, narrow beams

2) Plain Strain - DOFs – 2 / node $\{U_x, U_y$ (in-plane translations)}

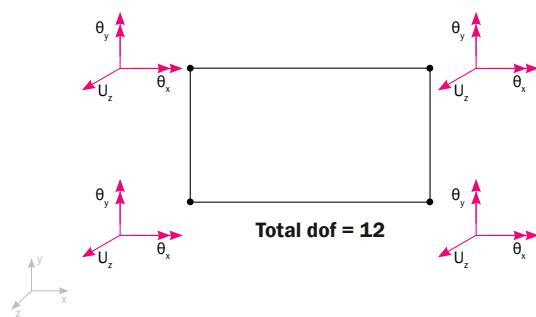
Strain in z direction (thickness) is zero ($\epsilon_z = 0$)



Practical Applications: Under ground pipes, wide beams, dams

Plane stress and plane strain elements are used for 2D (planar) problems.

3) Plate - DOFs – 3 / node $\{\theta_x, \theta_y$ (in plane rotations) + U_z (out of plane translation)

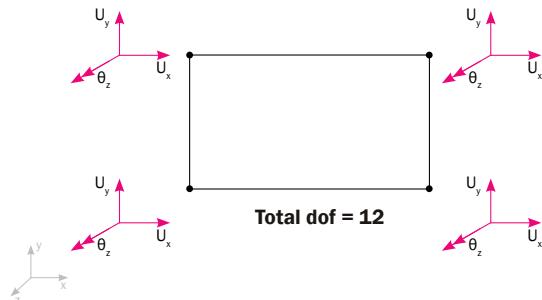


Practical Applications: Bending load application.

Plate elements are three- or four-node elements formulated in three-dimensional space. These elements are used to model and analyze objects such as pressure vessels, or structures such as automobile body parts. The out-of-plane rotational DOF is not considered for plate elements. You can apply the other rotational DOFs and all the translational DOFs as needed. Nodal forces, nodal moments (except when about an axis normal to the element face), pressures (normal to the element face), acceleration/gravity, centrifugal and thermal loads are supported. Surface-based loads (pressure, surface force, and so on, but not constraints)

and element properties (thickness, element normal coordinate, and so on) are applied to an entire plate element. Since these items are based on the surface number of the lines forming the element, and since each element could be composed of lines on four different surface numbers, how these items are applied depend on whether the mesh is created automatically (by either the mesher from a CAD model or the 2D mesh generation), or whether the mesh is created by hand. The surface number of the individual lines that form an element are combined as indicated in the table above to create a surface number for the whole element. Loads and properties are then applied to the entire element based on the element's surface number

4) Membrane - DOFs – 3 / node $\{U_x, U_y \text{ (in plane translations)} + \theta_z \text{ (out of plane rotation)}\}$

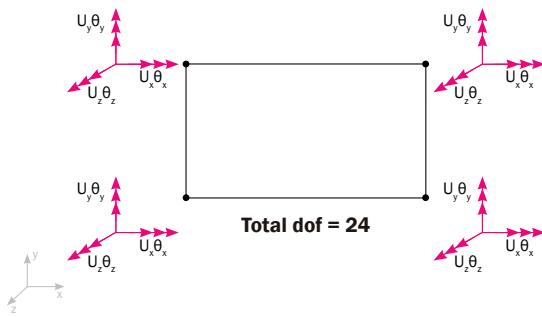


Practical Applications: Balloon, Baffles

5) Thin Shell - Thin shell elements are the most general type of element.

DOFs: 6 dof / node ($U_x, U_y, U_z, \theta_x, \theta_y, \theta_z$).

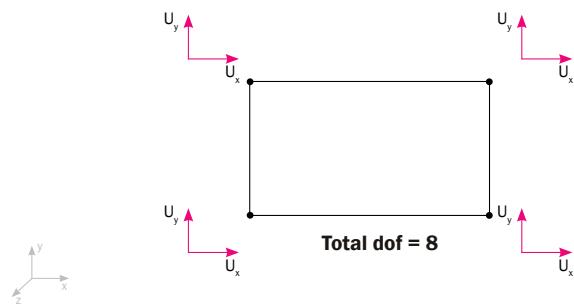
$$\begin{array}{lll} \text{Thin Shell} & = & \text{Plate} & + & \text{Membrane} \\ (\mathbf{U}_x, \mathbf{U}_y, \mathbf{U}_z, \theta_x, \theta_y, \theta_z) & = & \mathbf{U}_z, \theta_x, \theta_y & + & \mathbf{U}_x, \mathbf{U}_y, \theta_z \\ (3T+3R) & = & (1T+2R) & + & (2T+1R) \end{array}$$



Practical Application: Thin shell elements are the most commonly used elements.

6) Axisymmetric Solid - DOFs - 2 / node $\{U_x, U_z \text{ (2 in plane translations, Z axis is axis of rotation)}\}$

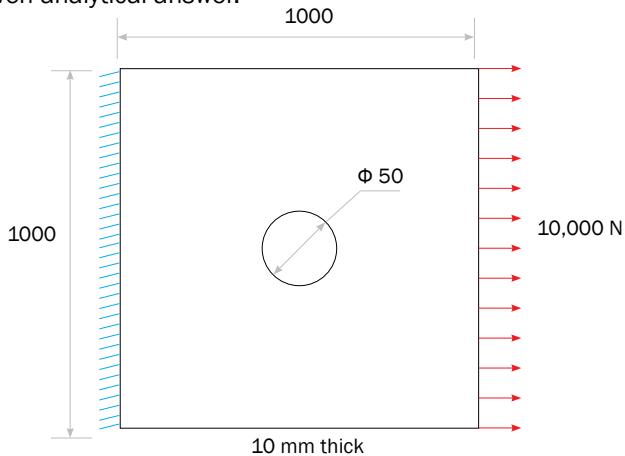
Why is the word ‘solid’ in the name of a 2D element? This is because though the elements are planar, they actually represent a solid. When generating a cylinder in CAD software, we define an axis of rotation and a rectangular cross section. Similarly, for an axi-symmetric model we need to define an axis of rotation and a cross section (planar mesh). The 2D planar mesh is mathematically equivalent to the 3D cylinder.



Practical Applications: Pressure vessels, objects of revolutions subjected to axi-symmetric boundary conditions

11.3 Thin Shell Elements

In the following we investigate the “performance” of quad and tria-elements by looking at a plate with a circular hole. The modeling results are then compared with a given analytical answer.



Analytical Answer

The Stress Concentration Factor (SCF) is defined as = max. stress / nominal stress

In this example the nominal stress is = $F/A = 10,000 \text{ N}/(1000 \text{ mm} * 10 \text{ mm}) = 1 \text{ N/mm}^2$

For an infinite plate SCF =3

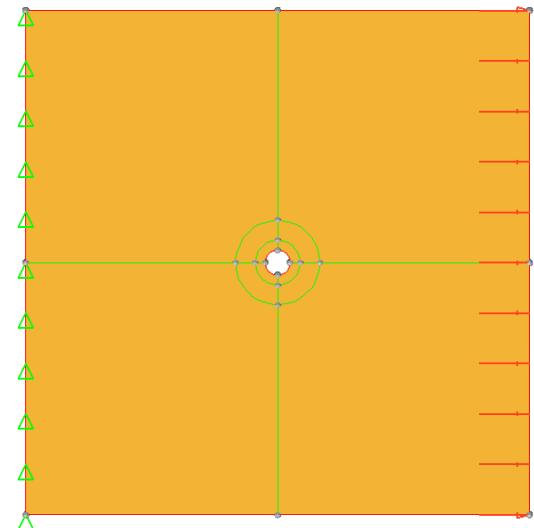
Hence, the maximum stress = Stress Concentration Factor (SCF) * nominal stress = 3 N/mm²

In the first part of this study the effects of element type (quad versus tria elements) on the modeling results are investigated. The global mesh size is 100.

The boundary conditions for all models are the same: the translational degrees of freedom (x-, y-, z- displacements=0) of all nodes along the left edge of the model are constrained (green symbols) whereas the nodes along the right edge are subjected to forces in the x-direction (total magnitude 10,000 N).

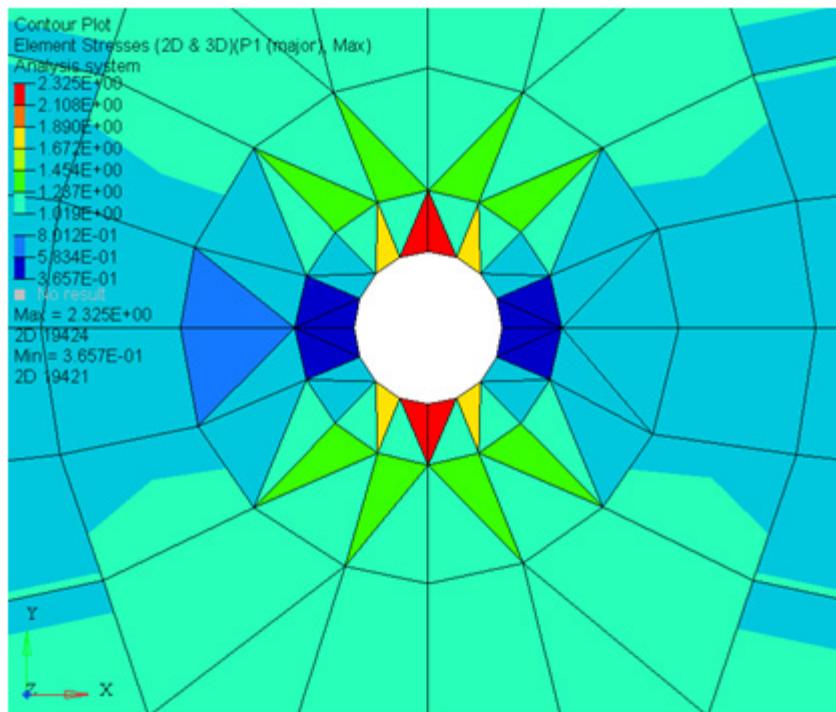
In order to better control the mesh pattern surrounding the hole, two so called “washers” have been introduced (i.e. the initial surface is trimmed by two circles with a radii of 45 mm and 84 mm, respectively).

In the stress contour plots shown below, the maximum principal stress is depicted, respectively. Note that by default the element stresses for shell (and solid elements) are output at the element center only. In other words, these stresses are not exactly the ones “existing” at the hole. To better resolve the stresses at the hole, the element stresses are output at the grid points using bilinear extrapolation (in HyperMesh activate the Control cards > Global output request > Stress > Location: Corner).

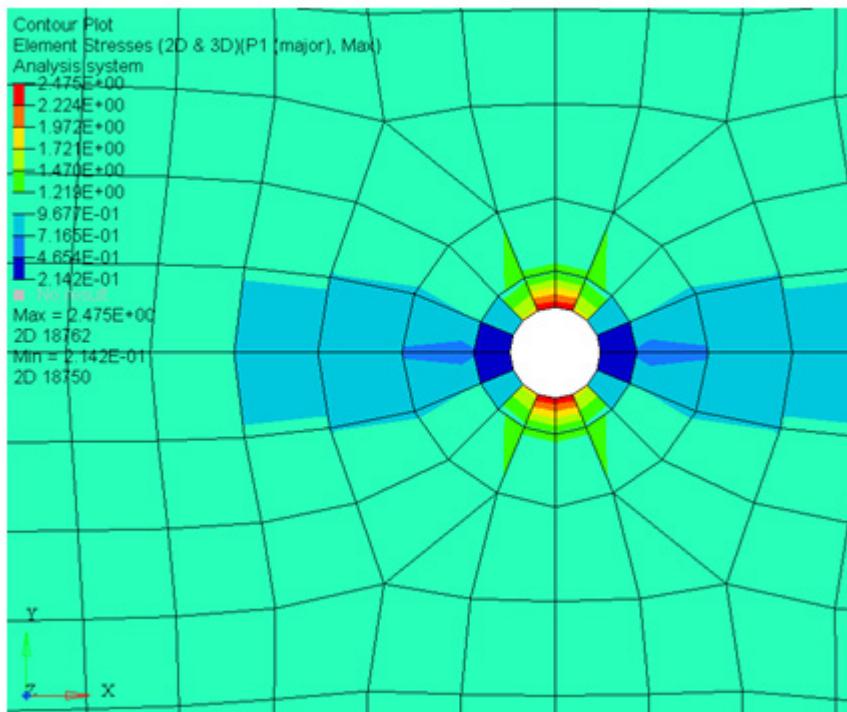


Effect Of Element Type (quads vs. trias)

Model 1: The hole is meshed with 16 tria elements. The maximum principal stress (corner location) is 2.32 N/mm² (the analytical result is 3 N/mm²).



Model 2: The hole is meshed with 16 quad elements. The maximum principal stress (corner location) is 2.47 N/mm² (the analytical result is 3 N/mm²).



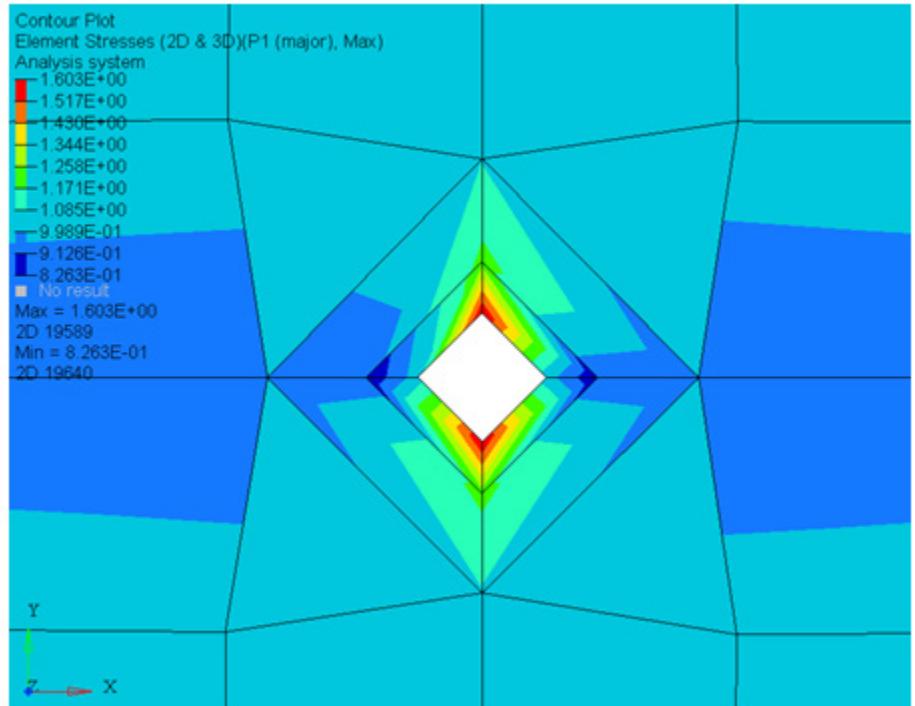
Despite the fact, that both results differ significantly from the analytical reference value of 3 N/mm^2 , it is apparent that the quad elements (17 % error) are “better” than tria elements (23 % error). All in all, both models are inappropriate when it comes to quantitatively assessing the stresses at the hole.

Moreover, another even more important lesson to be learned is that the FEM program does not tell you that the mesh is unacceptable – this decision is up to the CAE engineer

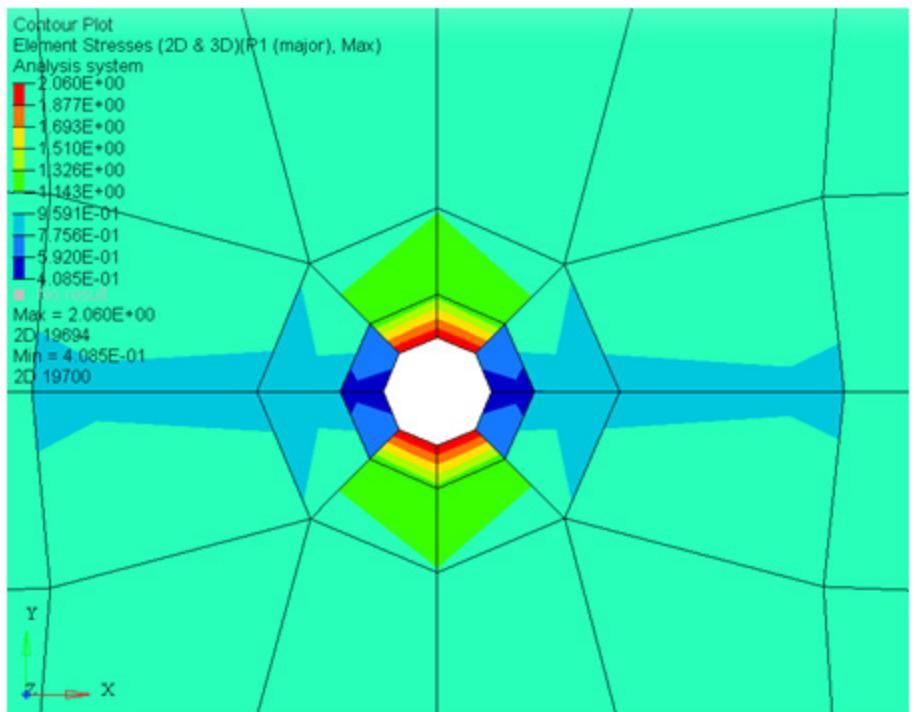
Effect Of Mesh Density

In the following the effect of element size, i.e. number of elements at the hole, on the modeling results is discussed.

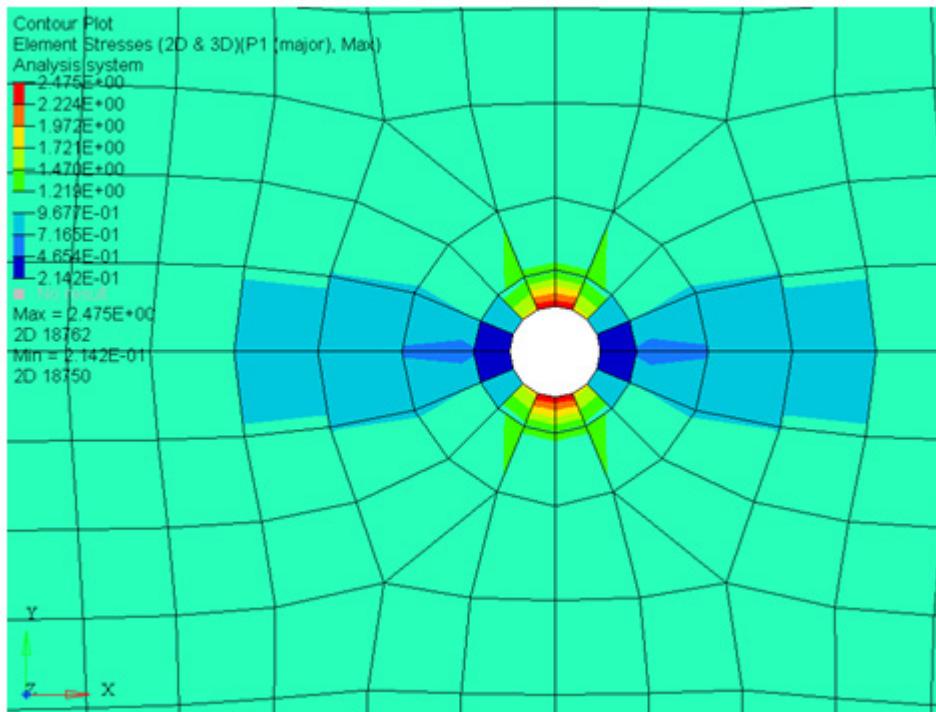
Model 3: The hole is meshed with 4 quad elements. The maximum principal stress (corner location) is 1.60 N/mm^2 (the analytical result is 3 N/mm^2).



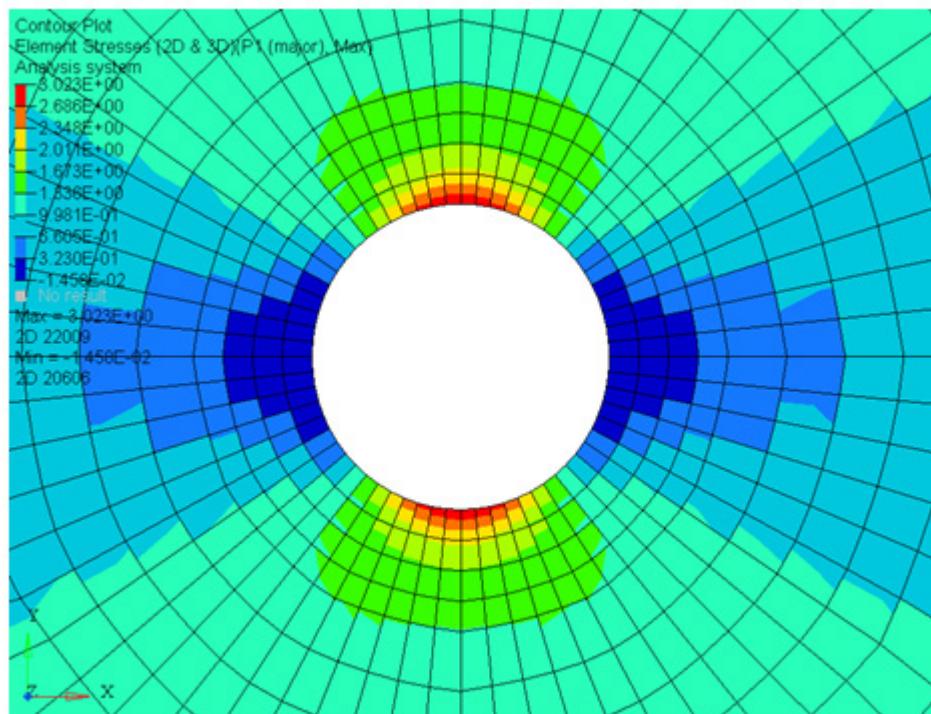
Model 4: The hole is meshed with 8 quad elements. The maximum principal stress (corner location) is 2.06 N/mm^2 (the analytical result is 3 N/mm^2).



Model 5: The hole is meshed with 16 quad elements. The maximum principal stress (corner location) is 2.47 N/mm² (the analytical result is 3 N/mm²).



Model 6: The hole is meshed with 64 quad elements. The maximum principal stress (corner location) is 3.02 N/mm² (the analytical result is 3 N/mm²).

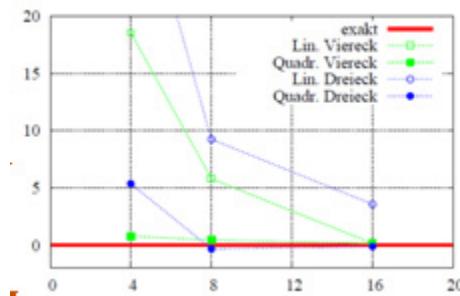


	Element Type	# Elements at Hole	Stress (N/mm ²) (corner location)
Model 1	Tria	16	2.32
Model 2	Quad	16	2.47
Model 3	Quad	4	1.6
Model 4	Quad	8	2.0
Model 5	Quad	16	2.47
Model 6	Quad	64	3.02

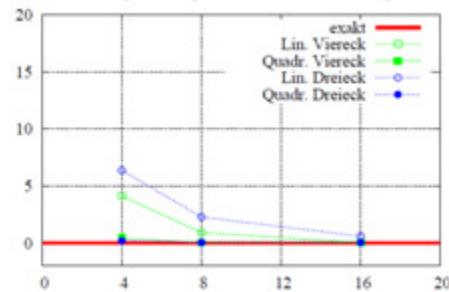
Conclusion:

The conclusion from the first exercise was that quad elements are better than triangular elements. As indicated by the results shown before: the greater the number of elements in the critical region (i.e. hole), the better its accuracy.

While we have been looking at linear elements (nodes at the element corners only), Professor Dieter Pahr (University Vienna, Austria) documented the differences between second order tria and quad elements. His study is also based on a plate with a hole. The image below is taken from his course notes: "Modeling, Verification and Assessment of FEM simulation Results".



In the figure above, the von Mises stress is depicted for linear and second order quad- and tria- elements. The vertical axes is error (%), and the horizontal axes is number of elements at 1/4 of the hole



In this image, the displacements at the hole is depicted for linear and second order quad- and tria- elements

Conclusion:

Second order quad elements behave/perform best, whereas linear trias are “problematic”. These element related effects are less severe while looking at displacements (nodal results).

If this is so, then why not always create a very fine mesh with the maximum possible number of nodes and elements? Why is the usual guideline for meshing 12-16 elements around holes in critical areas?

The reason is because the solution time is directly proportional to the (dof)². Also large size models are not easy to handle on the computer due to graphics card memory limitations. Analysts have to maintain a fine balance between the level of accuracy and the element size (dof) that can be handled satisfactorily with the available hardware configuration.

How Are Thumb Rules Made?

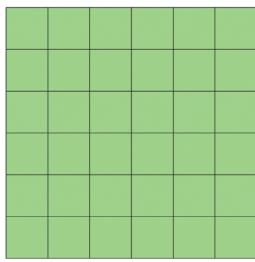
It is based on a simple exercise like the one above. The results of different mesh configurations are compared with a known analytical answer and the one which gives the logical accuracy with a reasonable solution time is selected.

Most of the industries follow the following thumb rule for the number of elements on holes:

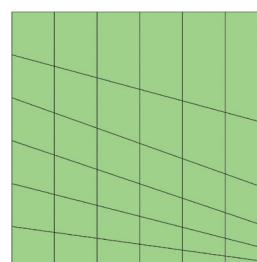
Minimum number of elements :

- critical areas = 12
- general areas = 6

11.4 Effect Of Biasing In The Critical Region

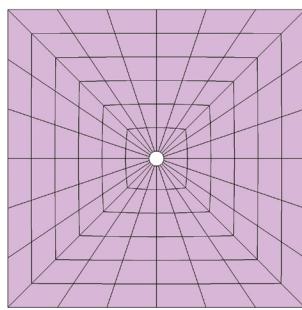


No bias

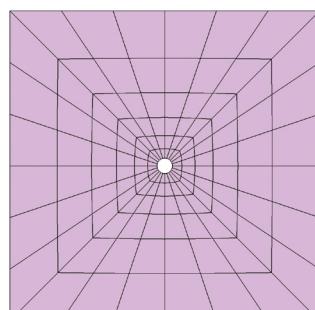


Bias 15 (on the right side)

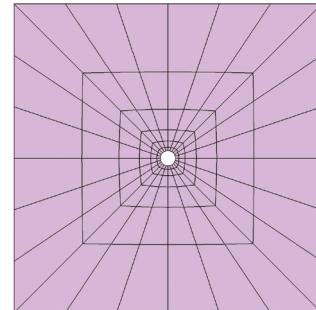
During meshing, an option named “biasing” may be used. When element length is not the same on the edge and biased towards a point, then it is known as biased meshing. Different commercial software calculate bias differently. One of the simplest schemes is where the bias factor is a ratio of the maximum element length divided by the minimum element length.



Bias 0

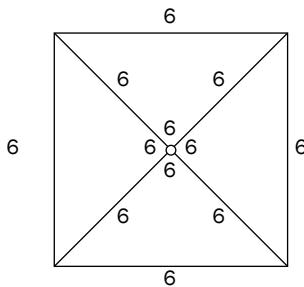


Bias 5



Bias 20

The above geometry was split along the diagonal and the bias was defined on the diagonals (at the edge point near the circular hole).



Biased Mesh Exercise:

Distribution of elements

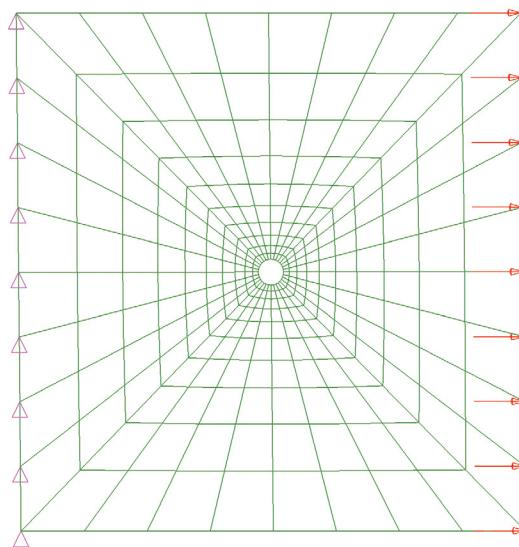
	Stress N/mm ² (exact answer 3 N/mm ²)	Nodes	Elements
No Bias	1.59	168	144
Bias 5	2.13	168	144
Bias 10	2.41	168	144
Bias 15	2.56	168	144
Bias 20	2.65	168	144

The conclusion from the previous example was that the higher the number of elements in the critical area, the higher the accuracy. The example on biasing shows that even without increasing the number of elements, one can achieve a better result just by the appropriate arrangement of the nodes and elements. This was done at no extra computational cost.

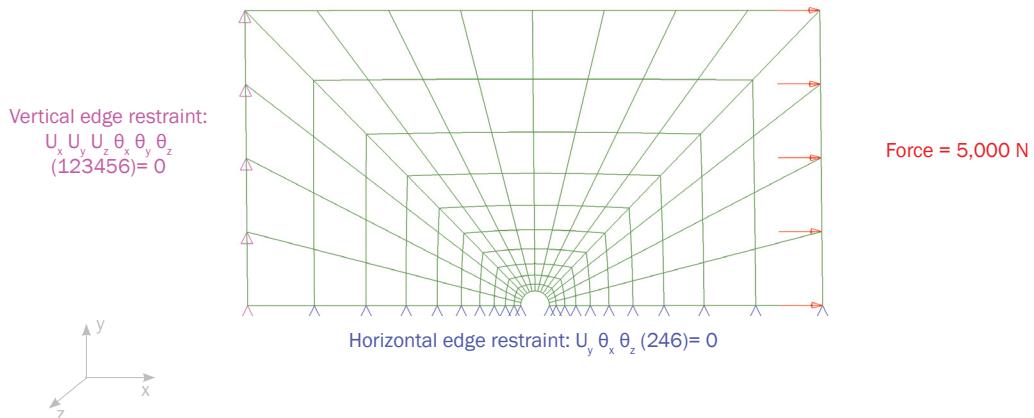
11.5 Symmetric Boundary Conditions

In order to reduce the total number of elements one may consider only 1/2 or 1/4 of the original structure. In this way a locally refined mesh (i.e. higher accuracy) may be used whereas the total number of elements may still be less than an evenly coarse meshed full model.

Full Plate:



Half Plate:



How to apply symmetric boundary conditions :

Step 1 : Write plane of symmetry for half plate i.e. x z

Step 2 : Fix in plane rotations (θ_x, θ_z) and out of plane translations (U_y).

Please find more details about symmetric boundary conditions in Chapter 10.

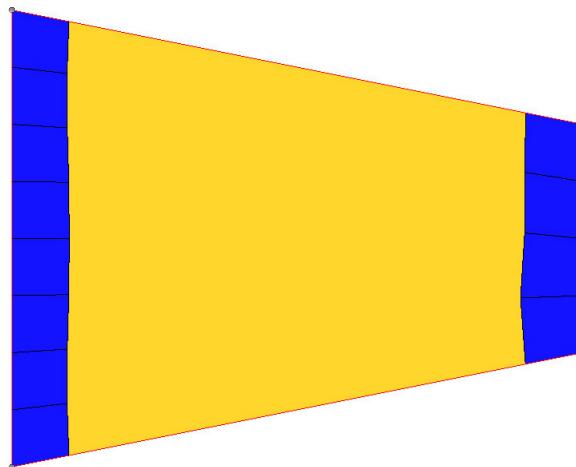
	Stress	Displacement	Nodes	Elements
Complete model	2.68	0.00486	352	320
Half Symmetry	2.68	0.00486	187	160

The advantage of symmetric boundary conditions is that the same accuracy is achieved at a lesser computational time and cost.

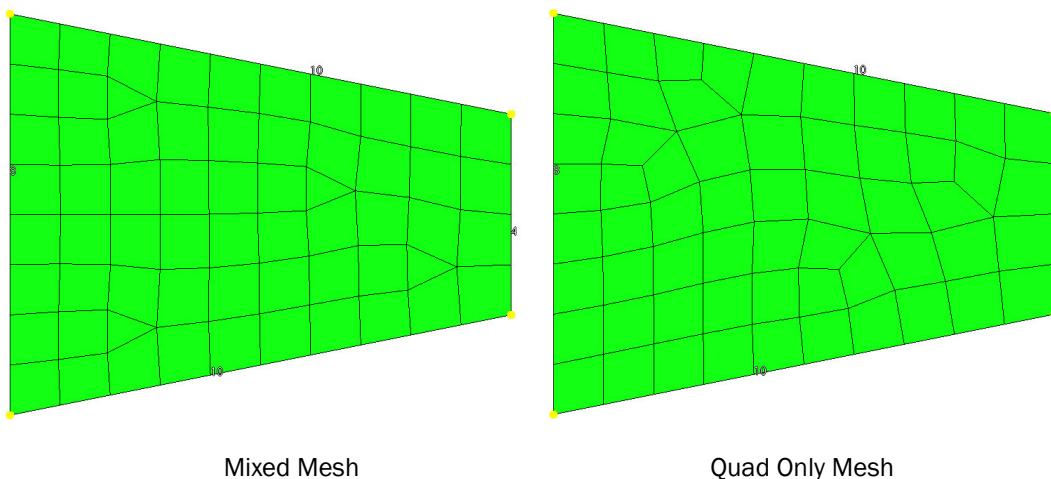
Symmetric boundary conditions should not be used for dynamic analysis (vibration analysis). It can not calculate anti nodes.

Different Element Type Options For Shell Meshing:

- 1) Pure quad elements
- 2) Mixed mode
- 3) Equilateral tria
- 4) (Right angle) R-tria



The image above depicts a geometry that has been partially meshed. How would you “link” both meshes? Depending on whether some single tria elements are allowed, you may use the mixed meshing method or the pure quad mesh. Both meshes are shown below. Note the more homogeneous mesh pattern resulting from “mixed” meshing.



The mixed mode element type is the most common element type used due to the better mesh pattern that it produces (restriction: total tria % < 5). Sometimes for structural analysis or for convergence and better results for a non linear analysis, the pure quadrilateral element meshing option is selected.

If Quads Are Better Than Trias, Why Not Always Mesh Using Only Quad Elements? Why Do FEA Software Provide The Option For Tria Elements?

1) Mesh transition: In structural and fatigue analysis, rather than a uniform mesh, what helps is a small element size in the critical areas and a coarse mesh or bigger elements in general areas. This type of mesh gives good accuracy with manageable dofs. Trias help in creating a smooth mesh transition from a dense mesh to a coarse mesh.

2) Complex geometry: Geometry features like rib ends or sharp cutouts demand for the use of triangular elements. If quads are used instead of trias, then it will result in poor quality elements.

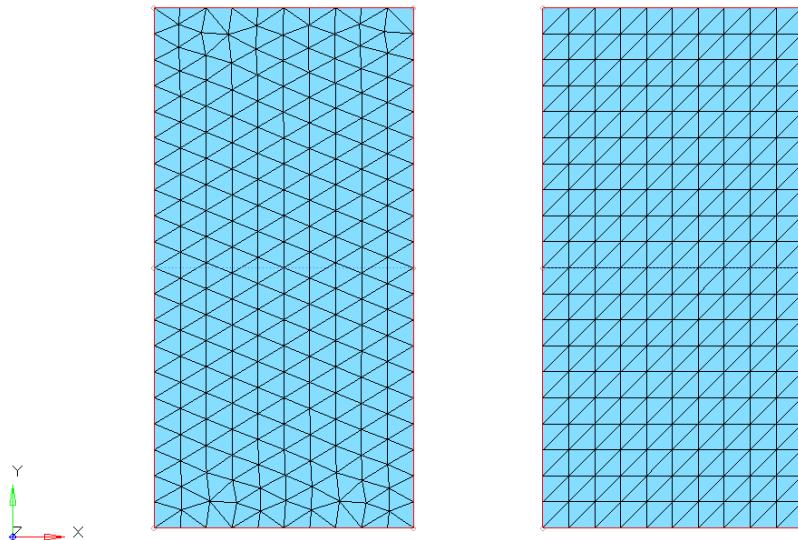
3) Better mesh flow: For crash or non linear analysis, systematic mesh flow lines where all the elements satisfy the required quality parameters is very important. Using a mix-mode element type instead of pure quad element type helps to achieve better flow lines and convergence of solution.

4) Tetra meshing (conversion from Tria to Tetra): For tetra meshing, all the outer surfaces are meshed using 2D triangular elements and then trias are converted to tetras. This methodology is discussed in detail in the next chapter.

5) Mold flow analysis: Mold flow analysis requires triangular elements.

Comparison Between Equilateral Tria And Right Angle Tria Meshing.

The default tria mesh in commercial software produce equilateral triangles while the R-tria option generates right angle triangles (generating a rectangular or square mesh and then splitting along the diagonal gives two trias per element).



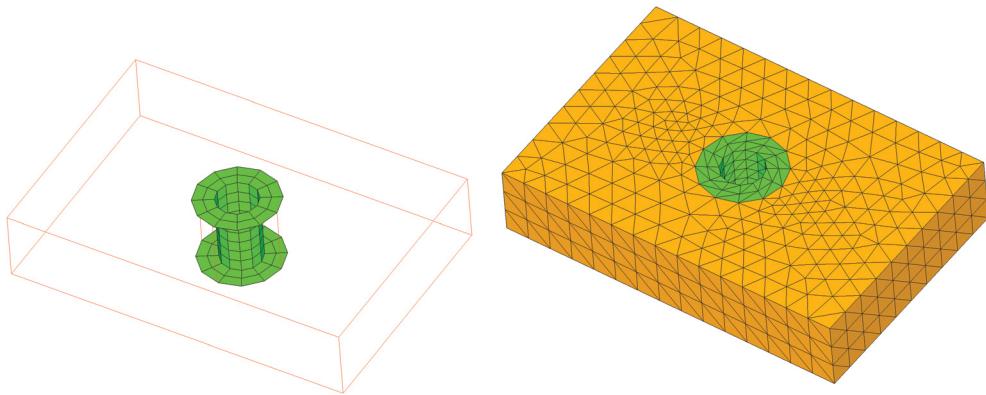
In the figure above, the left image shows meshing with equilateral trias, while the right image shows meshing with the option R-trias (which simply splits quad elements into two tria elements).

The ideal shape for a triangular element is an equilateral triangle and is theoretically better than a R-tria element. But for the following specific applications, R-trias have an advantage over equilateral trias.

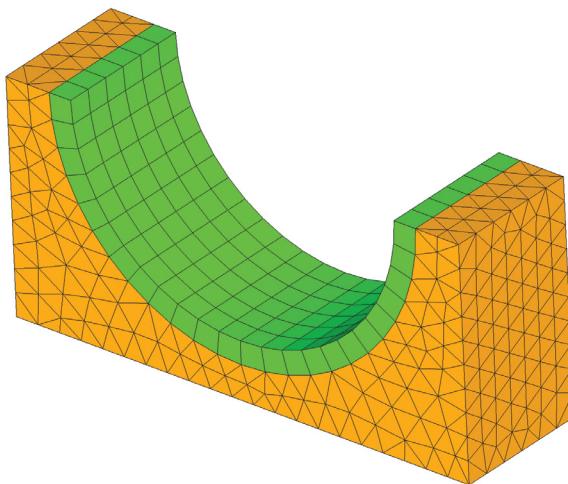
1) Tetra Meshing

For defining contacts, a similar mesh pattern on the two surfaces is desirable. The equilateral tria option produces a ziz-zag mesh and there is also no control over the mesh pattern. A similar mesh requirement could be achieved by generating a structured quadrilateral mesh (maintaining exactly same number of elements on two contact surfaces) and then splitting it to trias (R-tria). Typical applications are as follows:

a) Bolt hole and washer area:

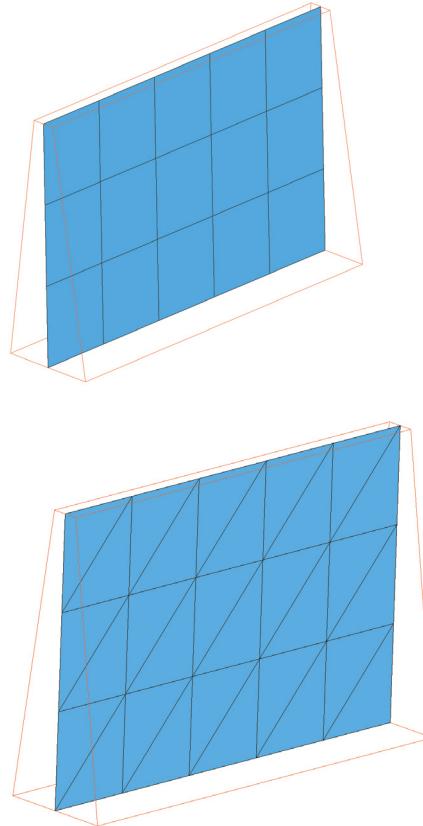
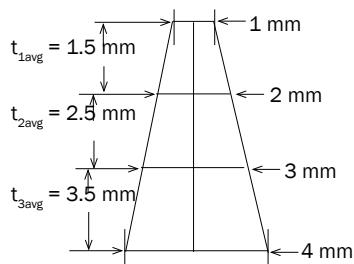


b) Bearing contact surfaces: Contact surfaces are meshed with quad elements (same mesh pattern and equal number of elements) and converted to trias before tetra conversion.



2) Variable Thickness Of Ribs For Mold Flow Analysis

The ribs are modelled using quad elements in three layers as shown below and then split to R-trias. The average section thickness is assigned to each different layer.

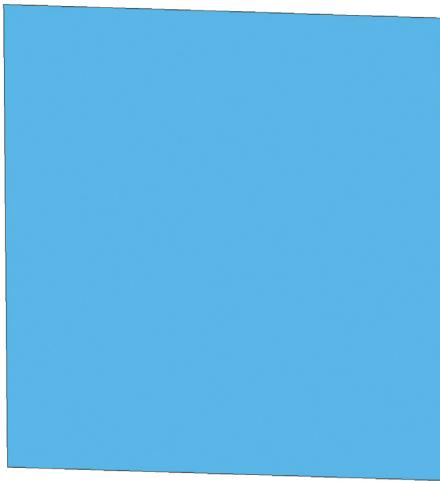


11.6 Geometry Associative Mesh

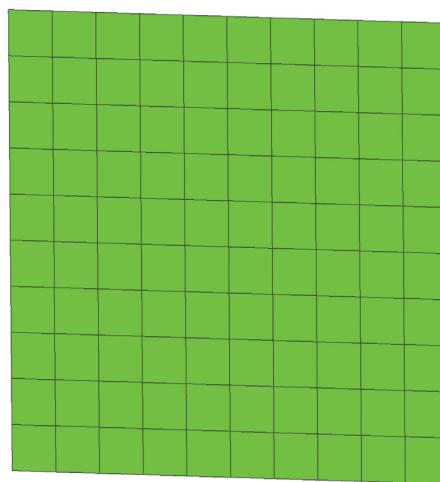
Creating a geometry associative mesh allows for automatic meshing to be carried out by picking surfaces or volumes from the geometry. The generated mesh is associative with the geometry.

Advantages:

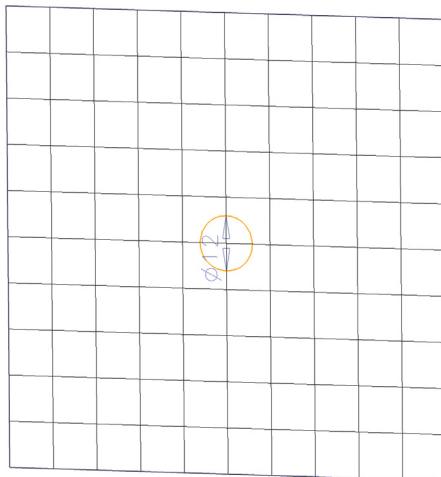
- 1) If the geometry is changed, then the mesh will also change automatically.
- 2) Boundary conditions could be applied on the geometry (edges, surfaces instead of nodes and elements, etc.) which is more user friendly.



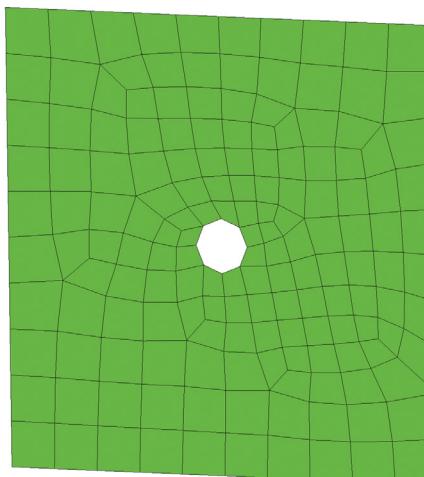
Original geometry



Geometry based mesh



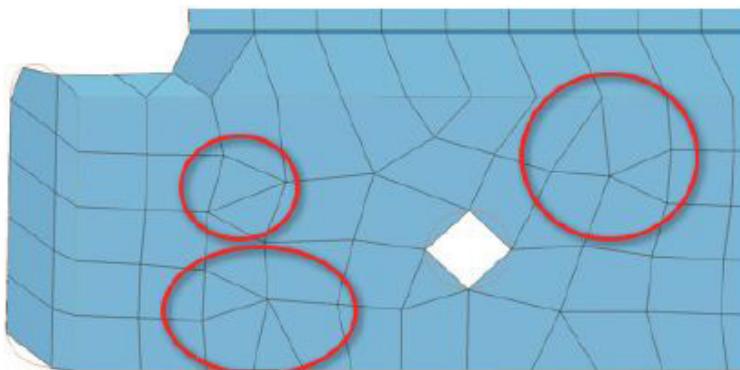
Geometry modification (cut hole at center)



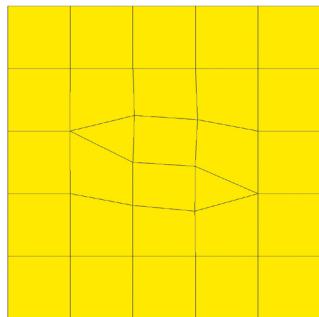
Auto update of mesh

11.7 How Not To Mesh

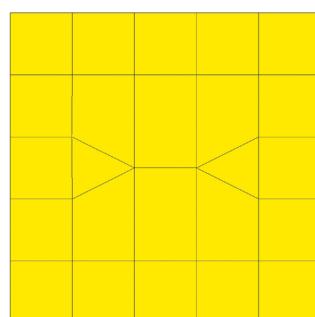
- 1) Back to back triangles should be avoided. Two tria elements should not be connected to each other directly.



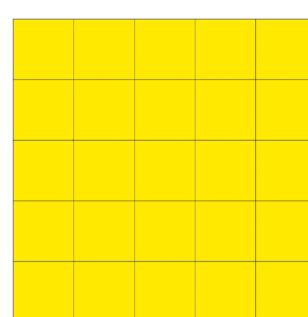
- 2) On plane surfaces triangular element should be avoided.



not recommended

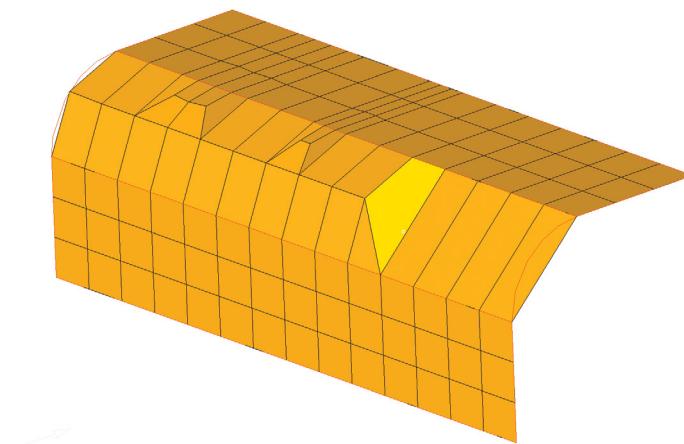


not recommended



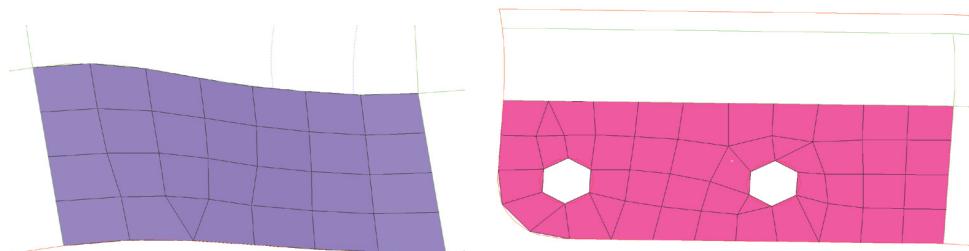
recommended

- 3) No mesh transition on constant radius fillets / curvatures

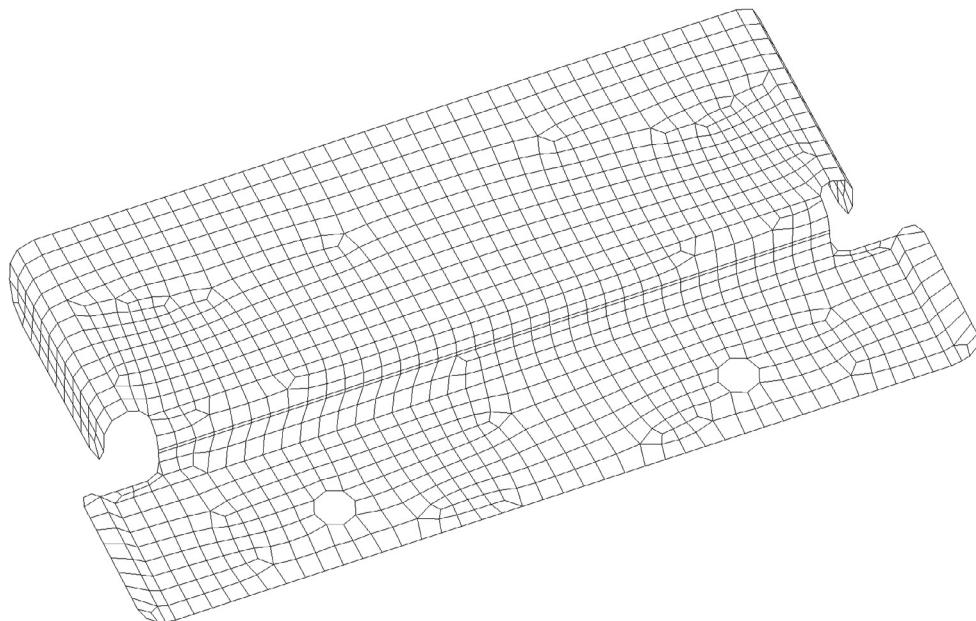


The mesh transition should be carried out on the planer surfaces (and not in the area of the fillet)

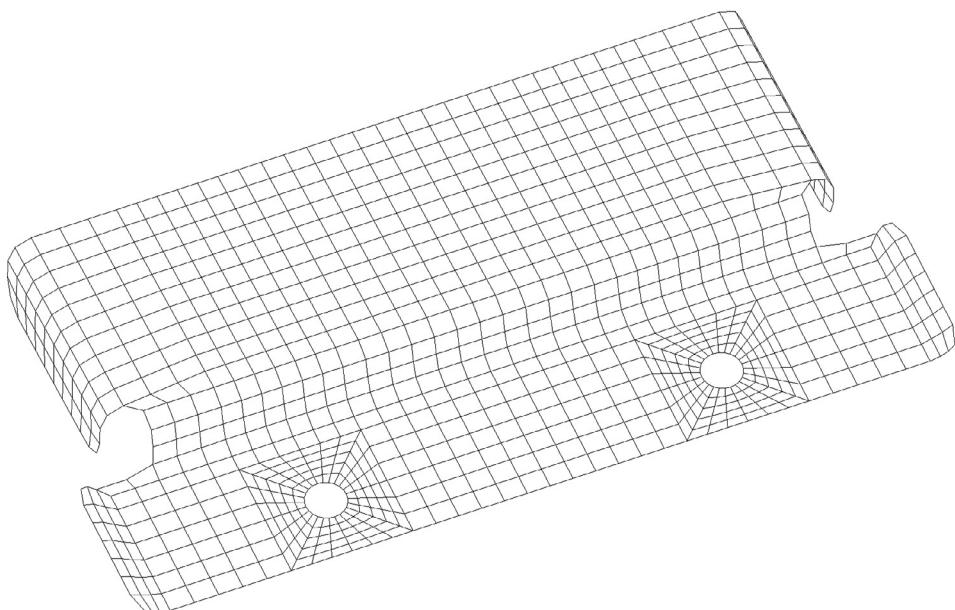
4) Avoid tria elements on outer edges or holes



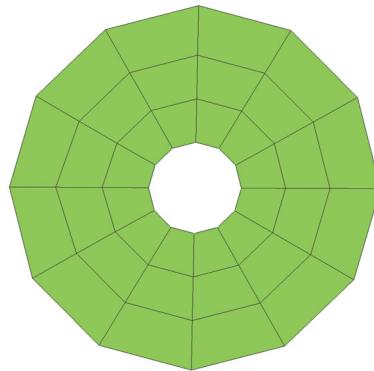
5) What is **not** acceptable at a professional level



What is acceptable.

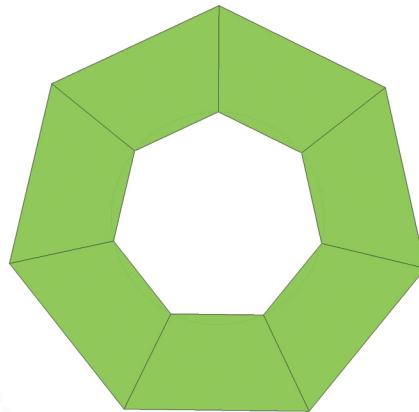


- 6) Circular holes should be modelled carefully with a washer (1.5 to 2 times diameter) and a minimum of two layers around the hole.



- 7) Holes should be modelled with an even number of equally spaced elements:

For a better representation of the hole geometry and smooth mesh flow lines, holes should be modelled with an even number of elements (like 6, 8, 12, 16 etc. rather than 5, 7, 9 or 13).

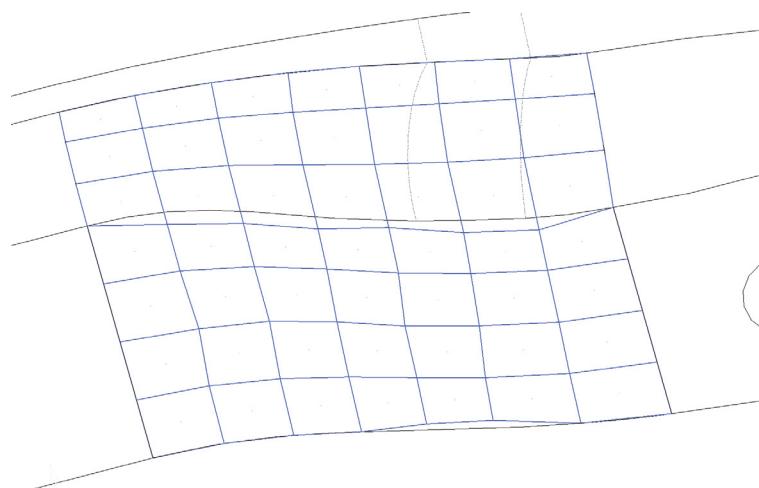


- 8) Nodes should lie properly on the surface, with no deviation (and no kinks).

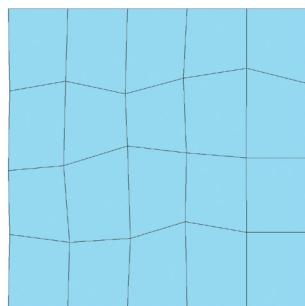


Switch off the element mesh lines and observe the contour (in particular at curvatures). Kinks as shown above are not acceptable.

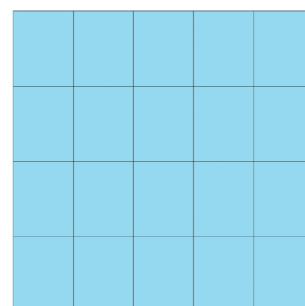
9) Follow the feature lines (nodes should lie exactly on the edges).



10) Instead of a zig-zag distribution, a structured or smooth mesh is recommended (nodes aligned in a straight line)



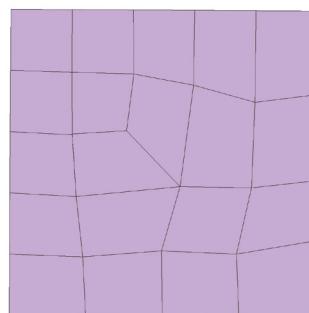
Not recommended



Recommended

Use of a “smooth” option, provided by most of the commercial software, helps in achieving systematic mesh.

11) For crash analysis, follow the mesh flow line requirement.



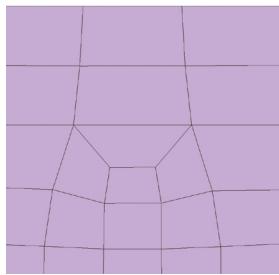
Diamond elements are not allowed

12) For crash analysis, rotating quads are not allowed.

Rotating quads

Recommended for structural analysis

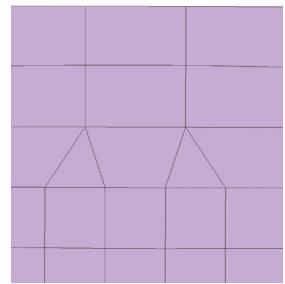
Not recommended for crash analysis



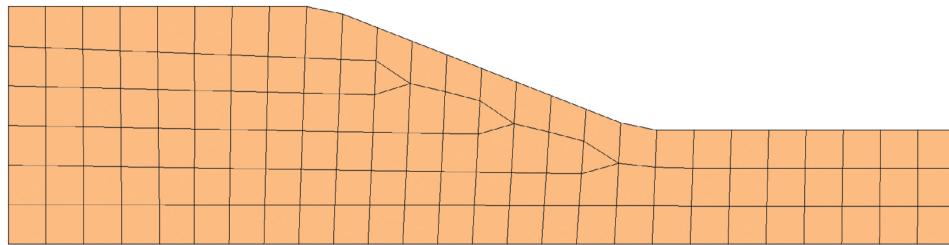
Recommended

Recommended for crash analysis

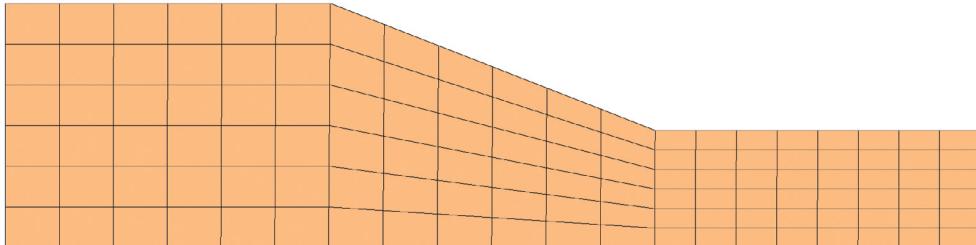
Not recommended for structural analysis



13) For crash analysis, constant mesh size (by using trias) is preferred (due to a minimum element length and a time step criteria).



Recommended for crash analysis



Variable mesh size not recommended for crash but recommended for structural analysis

11.8 Creating 2D Elements in HyperMesh

A surface mesh or “shell mesh” represents model parts that are relatively two-dimensional, such as sheet metal or a hollow plastic cowl or case. In addition, surface meshes placed on the outer faces of solid objects are used as a baseline mapping point when creating more complex 3D meshes (the quality of a 3D mesh largely depends on the quality of the 2D mesh from which it is generated).

Three-noded trias, four-noded quads, six-noded trias, and eight-noded quads can all be built in HyperMesh. These two-dimensional elements can be built in any of the following panels. A detailed look at automeshing and shrink wrap meshing will follow. For additional information regarding the other panels, please refer to the appropriate video.

Below is a listing of the panels available for creating and editing 2D elements. Most of these tools are located in the menu bar by selecting Mesh > Create > 2D Elements.

- *cones*: Builds elements on conic or cylindrical surfaces.
- *drag*: Builds elements by dragging a line, row of nodes, or group of elements along a vector.
- *edit element*: Builds elements by hand.

- *elem offset*: Builds elements by offsetting a group of elements in the direction of their normals.
- *line drag*: Builds elements by dragging a line or group of elements along or about a control line.
- *planes*: Builds elements on square or trimmed planar surfaces.
- *ruled*: Builds elements between two rows of nodes, a row of nodes and a line, or two lines.
- *spheres*: Builds elements on spherical surfaces.
- *spin*: Builds elements by spinning a line, row of nodes, or group of elements about a vector.
- *spline*: Builds elements that lie on a surface defined by lines.
- *torus*: Builds elements on toroidal surfaces

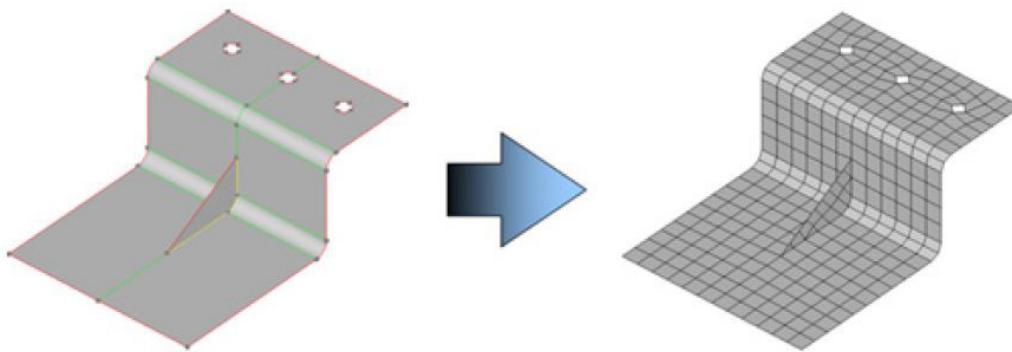
- *automesh*: Builds elements on surfaces according to user specifications (further details given below).
- *shrink wrap*: Builds 2D (optionally 3D) simplified meshes of existing complex models. Further details are given further below and in the chapter about 3D (solid) meshing.

11.9 Automeshing

The Automesh panel is a key meshing tool in HyperMesh. Its meshing module allows you to specify and control element size, density, type, and node spacing, and also perform quality checks before accepting the final mesh.



The optimal starting point for creating a shell mesh for a part is to have geometry surfaces defining the part. The most efficient method for creating a mesh representing the part includes using the Automesh panel and creating a mesh directly on the part's surfaces. The Automesh panel can be accessed from the menu bar by selecting Mesh > Create > 2D AutoMesh. A part can be meshed all at once or in portions. To mesh a part all at once, it may be advantageous to first perform geometry cleanup (please see Chapter 3 on Geometry) of the surfaces, which can be done in HyperMesh.

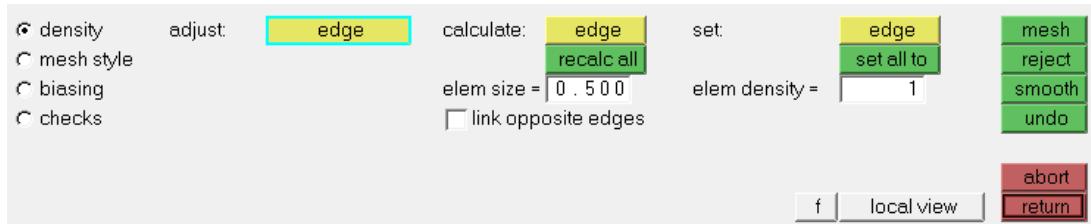


Provided your geometry is clean e.g. the surface of the rib is merged with the adjacent surfaces, then the resulting mesh will be automatically compatible (all elements are connected with each other).

There are two approaches to the Automesh panel, depending on whether or not you use surfaces as the basis for the operation.

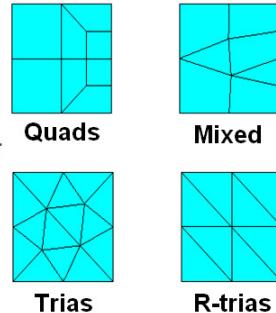
1. If you use surfaces, you may choose from a greater variety of algorithms, have more flexibility in specifying the algorithm parameters, and employ the mesh-smoothing operation to improve element quality.
2. If you do not use surfaces, the meshing process is usually faster and uses less memory. Most of the functions are still available and operate in the same way. Furthermore, there are situations in which it is not possible or not desirable to create a surface.

For either method, the module operates the same. You interactively control the number of elements on each edge or side and can determine immediately the nodes that are used to create the mesh. You can adjust the node biasing on each edge to force more elements to be created near one end than near the other, which allows you to see immediately the locations of the new nodes.



You can also specify whether the new elements should be quads, trias, or mixed and whether they should be first or second order elements.

- quads – Quad dominant mesh; will use trias if necessary
- trias – Equilateral (60-60-60) tria mesh
- mixed – Quad mesh with use of trias to create as rectilinear a mesh as possible
- R-trias – Isosceles (45-45-90) tria mesh
- quads only – Only quad elements created



The created mesh can also be previewed, which allows you to evaluate it for element quality before choosing to store it in the HyperMesh database. While you are in the meshing module, you can use any of viewing tools on the Visualization toolbar to simplify the visualization of complex structures in your model.

If you use surfaces, you can specify the mesh generation and visualization options to use on each individual surface. You may choose from several mesh generation algorithms. Mesh smoothing is also available and you may select the algorithm for that operation as well.

What You Need To Know/Remember:

While working with the Automesh panel you will come across the following options and settings:

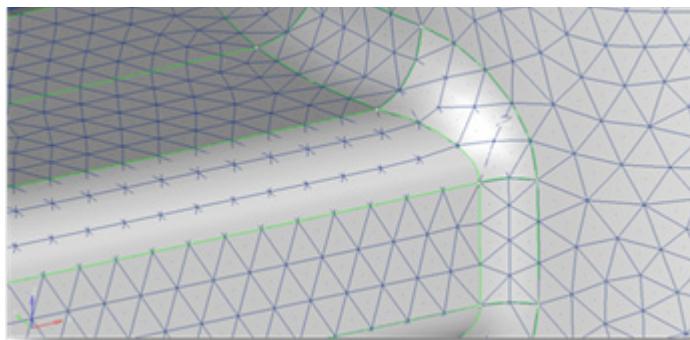
Element size = the element size in the model may deviate from the specified size considerably (it depends on the size of the surface).

Mesh type = mixed; default (is a combination of many quad-shaped elements and some tria elements). Leads to rather smooth meshes.

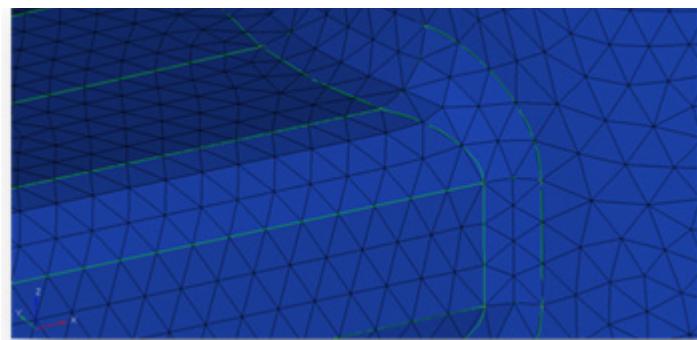
Elems to surf comp vs. Elems to current comp = specifies the “storage” place of the elements.

Start meshing, explore the meaning of the other settings later! What may happen is that the mesh looks a bit weird ...

Some surfaces apparently cause trouble. This may not actually be a problem, but a matter of your visual settings. In this example, the geometry is still shaded, overprinting the mesh in some spots. Displaying the geometry in wireframe and shading the elements will improve the mesh “visibility”.

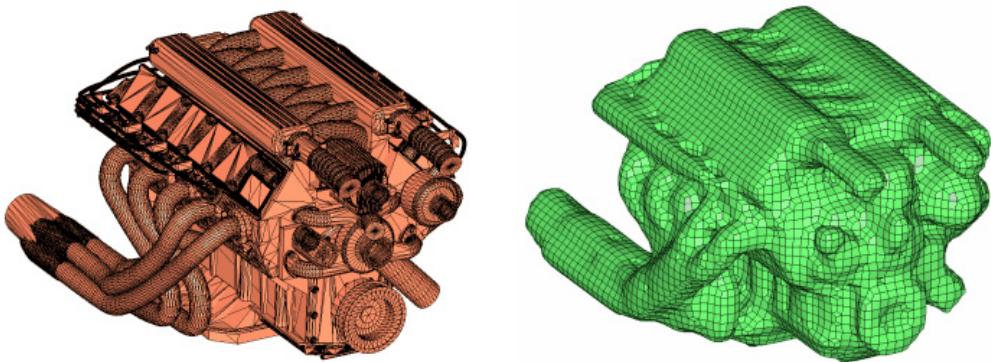


Note: In case you don't see any mesh, check the Model Browser and the status of the corresponding collector (is the elem icon activated?)



11.10 Shrink Wrap Meshing

Shrink wrap meshing is a method to create a simplified mesh of a complex model when high-precision models are not necessary, as is the case for powertrain components during crash analysis. The model's size, mass, and general shape remains, but the surface features and details are simplified, which can result in faster analysis computation. You can determine the level of detail retained by determining the mesh size to use, among other options.



You can shrink wrap elements, components, surfaces, or solids.

The shrink wrap allows for wrapping of multiple components if they are selected.

The selection provides the option to wrap all elements, components, surfaces or solids, or only a certain portion of the model if desired. The input to the shrink wrap (that is, the model parts that you wish to wrap) can consist of 2D or 3D elements along with surfaces or solids.

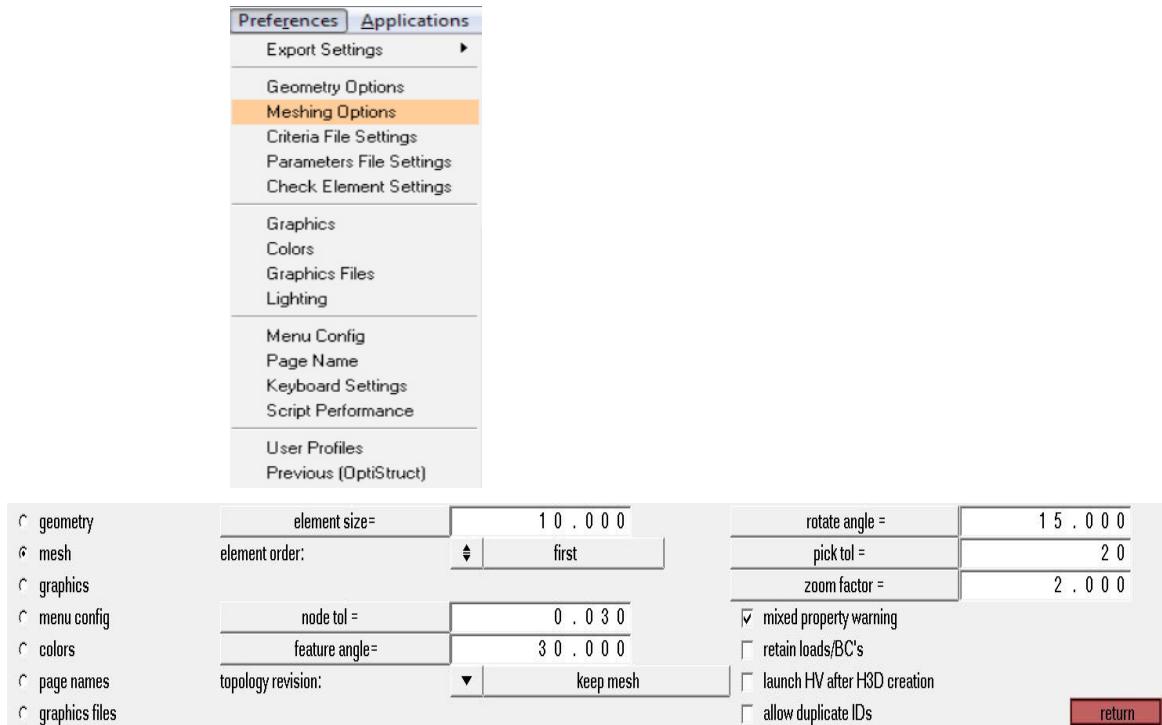
A shrink wrap mesh can be generated as a surface mesh (using a loose or tight wrapping), or as a full-volume hex mesh by use

of the Shrink Wrap panel. This panel is located in the menu bar by selecting Mesh > Create > Shrink Wrap Mesh. The distinction between surface or volume mesh is an option labeled generate solid mesh. The Shrink Wrap panel is covered in more detail in the 3D meshing section.

11.11 Meshing FAQ's

1. What To Do When You Are Unable To Specify A Minimum Elem Size Lesser Than A Particular Value In Automesh Panel?

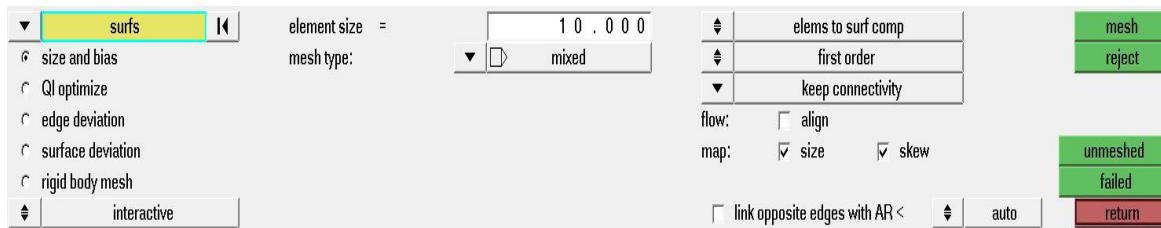
In the menu, go to 'Preferences' > 'Meshing options' panel, or press the <o> key.



Specify the element size in the element size field, this specifies a default edge length for elements, in the same units that the loaded model was created in (mm, inches, etc). This determines the default values to be used for meshing, such as in the automesh panel.

2. What To Do When You Get The Error One Or More Surfaces Failed To Mesh? How To Retrieve The Failed Surfaces From Meshing?

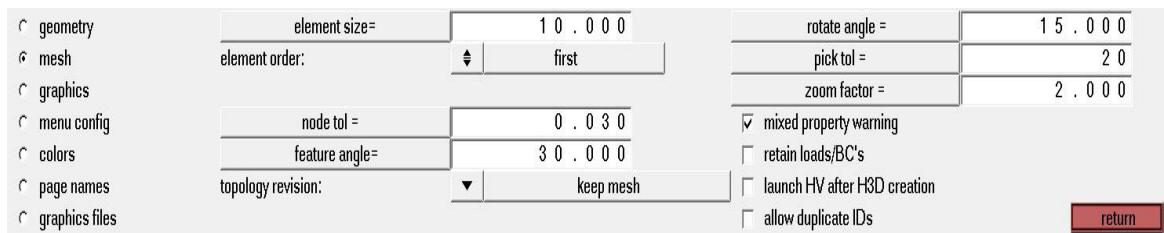
In 2D > automesh > there are two options: 1. failed surfaces, 2. unmched surfaces. Click on one of the options accord-



ing to whether you search for failed or unmched surfaces. The corresponding surfaces will get highlighted. You can save these surfaces into the so-called "user mark" (click on the yellow panel "surfs" will open the extended selection menu, then select the option "save"). Later, for instance in the "mask" panel the saved surfaces can be "retrieved" from within the extended selection menu. By inverting the selection (because the failed surfaces are active) all other surfaces will be masked after executing the mask command.

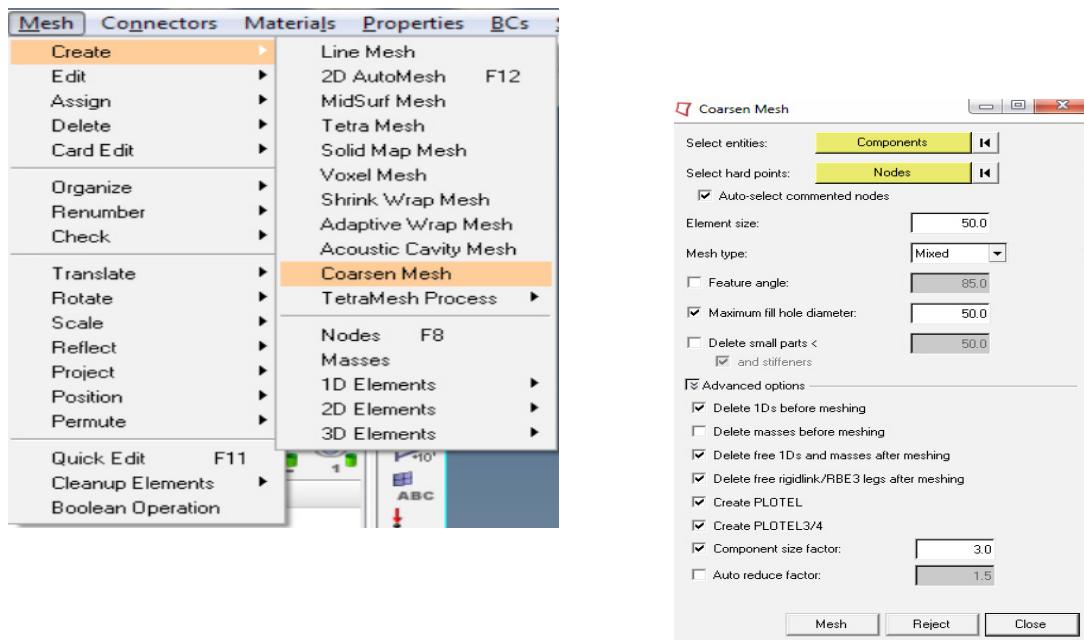
3. How To Edit A Surface But Not Have The Mesh Change?

In the menu, go to ‘Preferences’ > ‘Meshing options’ panel, and for ‘topology revision’ choose ‘keep mesh’.



4. How To Coarsen Your 2D Mesh?

Go to Mesh menu > create > coarsen mesh



Use the Coarsen Mesh utility to simplify the mesh by combining many small elements into a smaller number of larger ones. This dialog allows you to pick the Components that you wish to simplify, any Hard points (such as those defining a hole or ridge) that must be preserved, a new element size, and a mesh type (Mixed or Trias-only). In both cases, you must click the selector twice, as if you were accessing its extended entity selection menu; however, the second click opens a temporary panel in the panel area. This panel allows you to select the desired components or nodes, and then proceed in order to close the panel and return to the Coarsen Mesh dialog. Once you set the desired options, you can “Mesh” the selected components. If the results are not satisfactory, you can “Reject” the new coarse mesh, change the options, and try again.

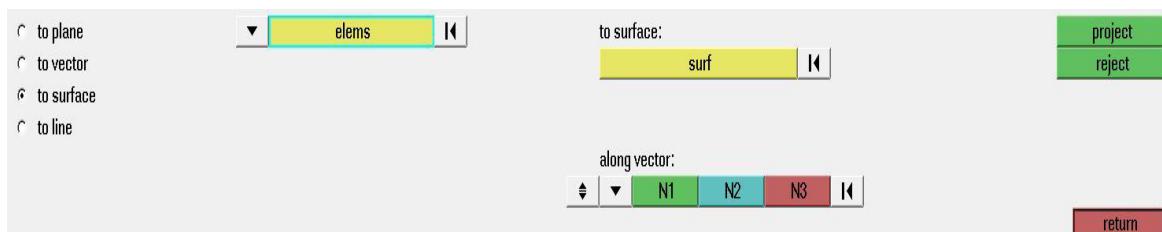
Manually

Detach elements from its neighbouring mesh and remesh the selection can be a bad idea if a good connectivity is required or the number of elements is too large.

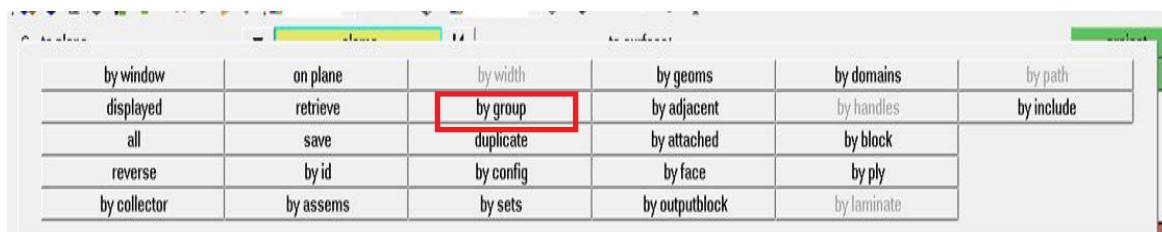
Create surface from elements, delete the selected elements and remesh the surface, which has points on its edges created from the surrounding mesh. Based on the difference of old and new mesh sizes, can provide a very good connectivity

5. How To Create A Matching Mesh On Two Opposing Identical Surfaces Or How To Project Elements On To Surface?

Tool page >project->to surface,

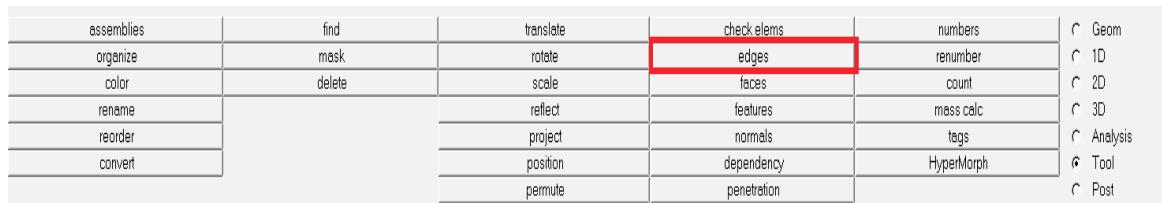


You may need to duplicate the existing elements so that a copy of the elements are projected.

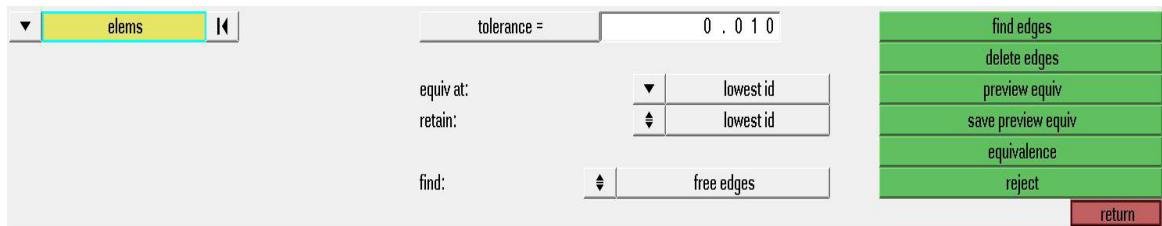


6. How To Delete Duplicate Nodes?

Go to Tools > edges



Select elements or components > select the elements or components using the extended entity selector > preview equivalence > if the automatic selection based on the tolerance specified is satisfactory > click equivalence.



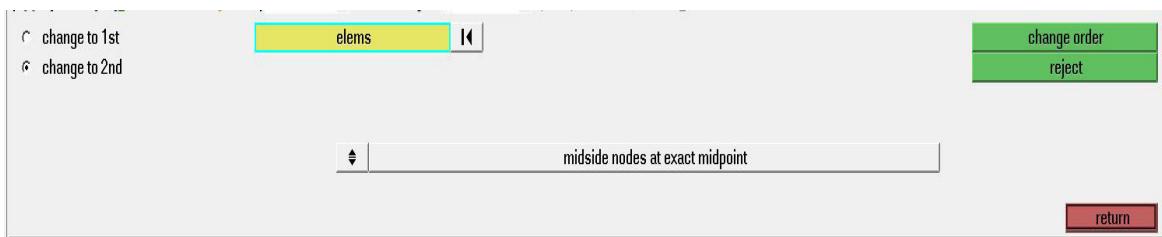
7. How To Make Changes To The Configuration Of Existing Elements? For Instance How To Change A 1D Bar Element To A Beam Element?

Go to 1D > config edit > select existing elements > select the necessary config and click on switch.



8. How To Change The Order Of Your Elements From First Order To Second Order Elements?

Go to 2D > order change > select elements and click on change order



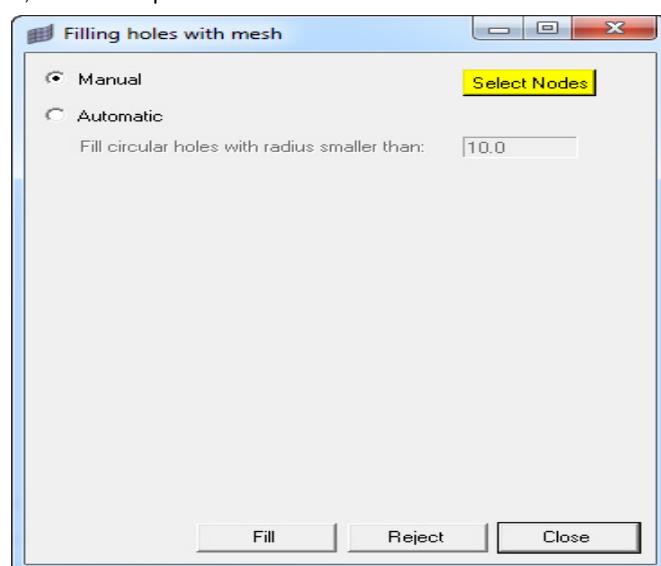
9. How To Fill One Or More Holes In Your Geometry With Automatically-Generated Mesh?

Go to utility tab > Geom/mesh > fill hole

Geom Tools	
Preserve Edges	
Project Points	
Isolate Surface	
ThinSolid=>Midsurf	
Washer	
Adj Circ Pts	

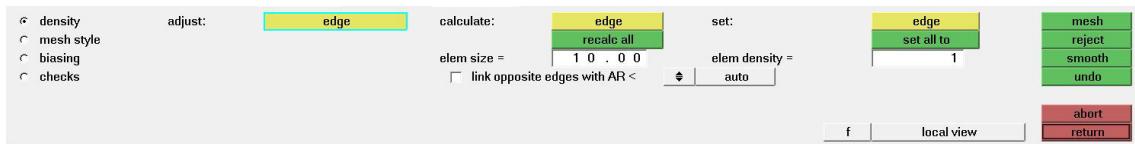
Mesh Tools	
Auto Connectors...	
Midsurf Thickness...	
Quick TetraMesh...	
Fix 2nd Order Midnodes	
Tetra Mesh Optimization	
Add Washer	
Periodic Mesh	
Trim Hole	Fill Hole
Box Trim	Bead

Summary	
FEA	Opti
Geom/Mesh	User
Disp	QA/Model

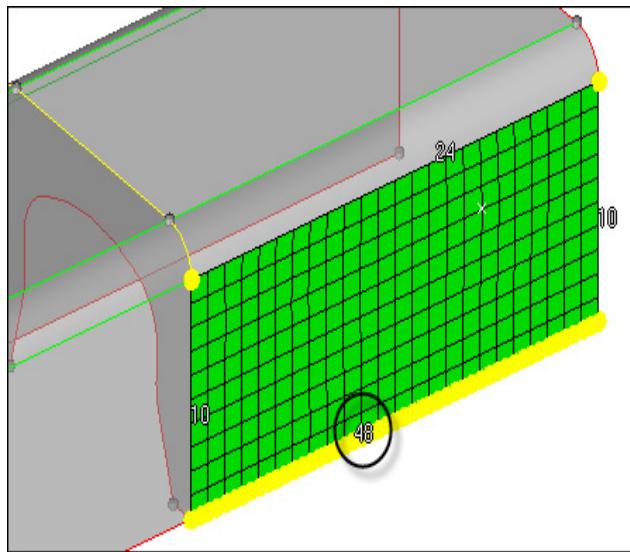


10. How To Specify A New Element Density Along Surface Edges?

Activating the “mesh” option in the “2D->automesh” panel opens up the density-mesh style-biasing-checks subpanel. In the density subpanel, click the selector, adjust: “edge” to make it active.



From the graphics area, left-click an edge's element density number to increase it by one.



Right-click an edge's number to decrease it by one. Click and hold the mouse pointer on an edge's number and drag the mouse up or down to increase or decrease the number.

Click mesh to update the preview mesh based on the change.

11.12 2D Meshing Tutorials And Videos

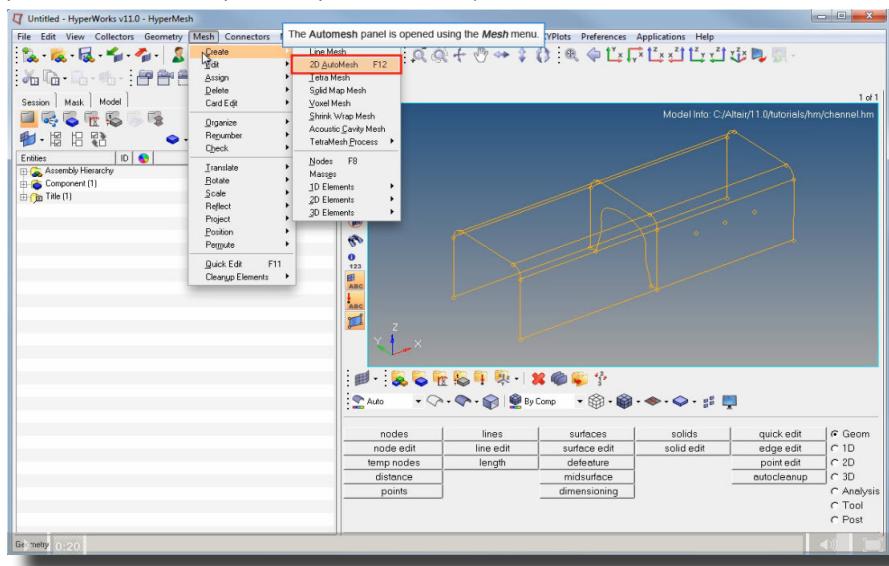
Recommended Tutorials:

The following tutorials are part of the HyperWorks installation (check the “help-documentation” for details):

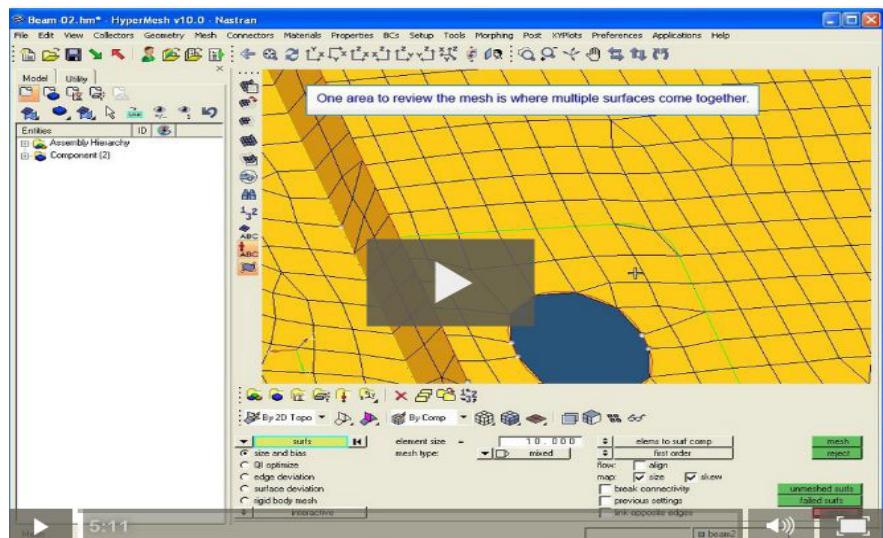
- HM-3100: AutoMeshing
- HM-3110: Meshing without Surfaces
- HM-3120: 2D Mesh in Curved Surfaces
- HM-3130: QI Mesh Creation
- HM-3140: Batch Meshing
- HM-3150: Meshing a Model Using Shrink Wrap

Recommended Videos

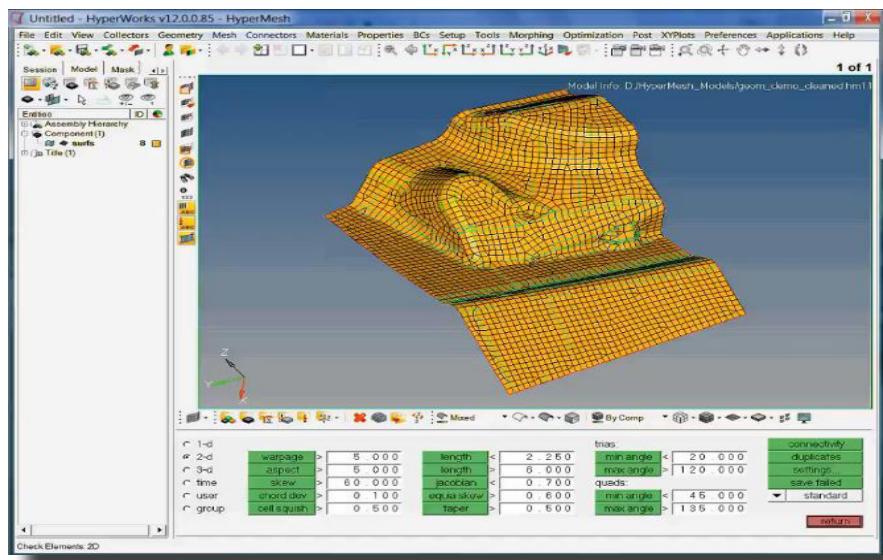
- **Automeshing** (<http://altair-2.wistia.com/medias/f555xw0boc>)



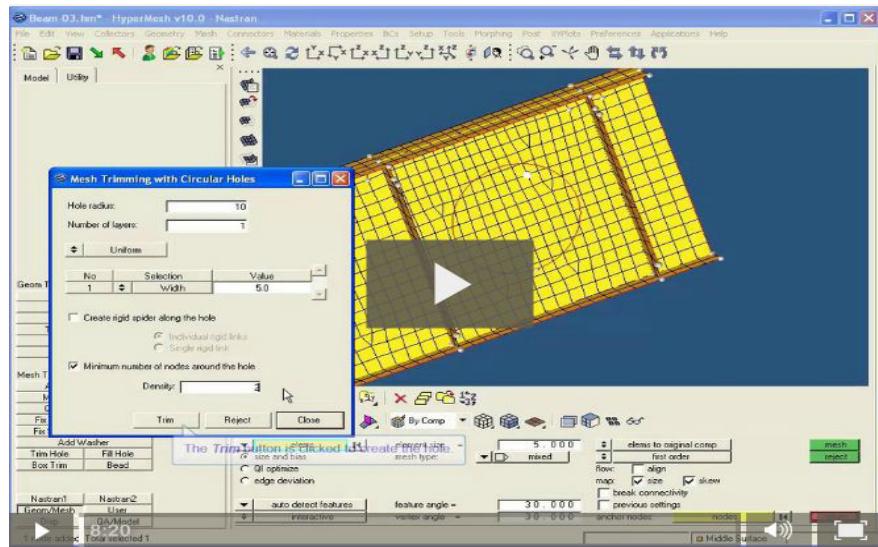
- **Mesh creation demonstration** (<http://altair-2.wistia.com/medias/tn4gdxmjde>)



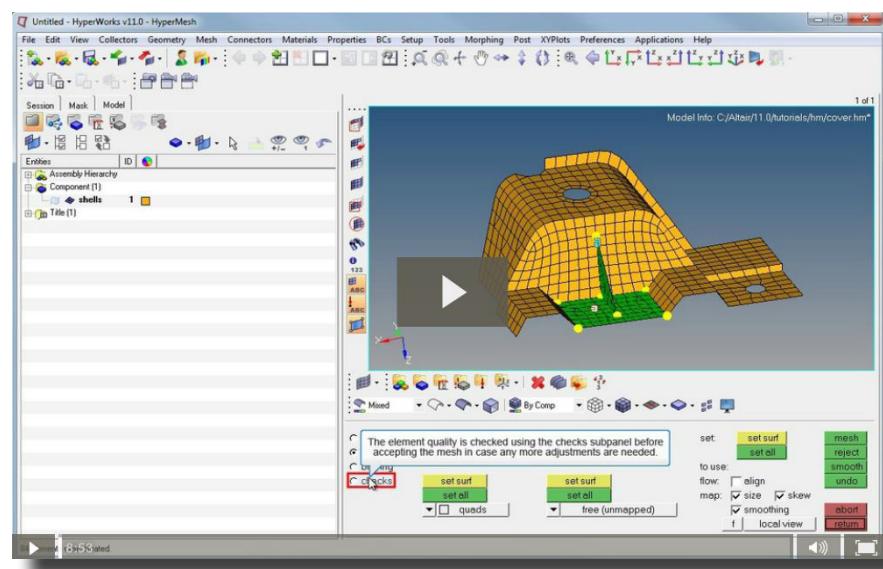
- **Mesh quality** (<http://altair-2.wistia.com/medias/dog2q02070>)



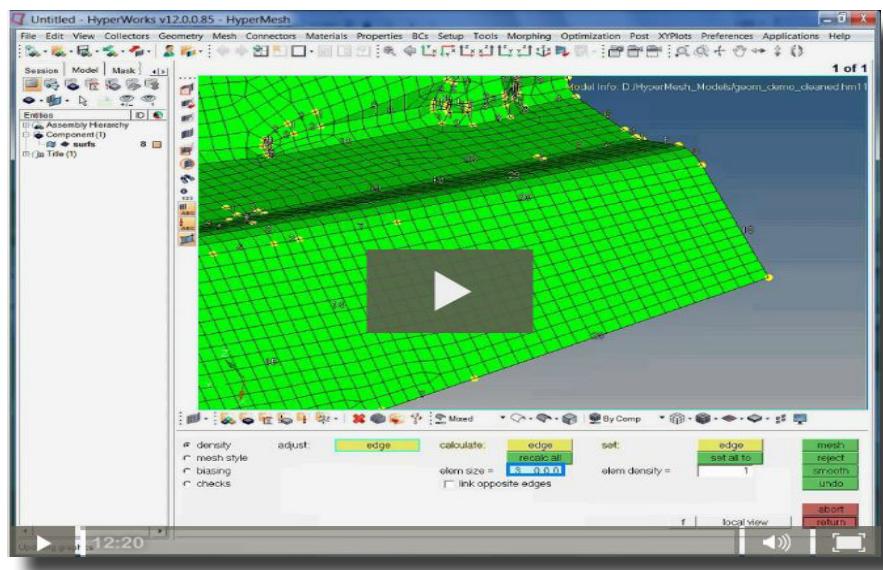
- **Editing mesh** (<http://altair-2.wistia.com/medias/1otl1kftp7>)



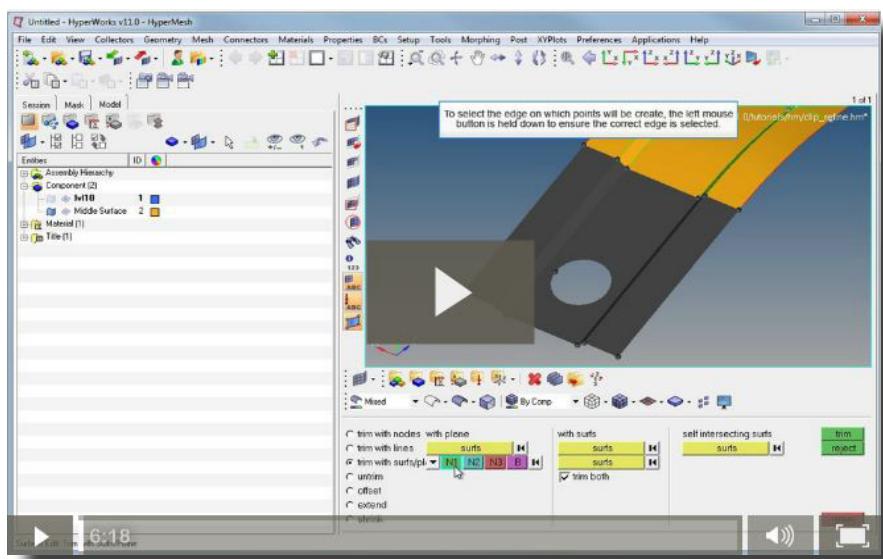
- **Mesh checking editing** (<http://altair-2.wistia.com/medias/ftu42udf77>)



- **Introduction in meshing** (<http://altair-2.wistia.com/medias/ujt7sb9sgm>)



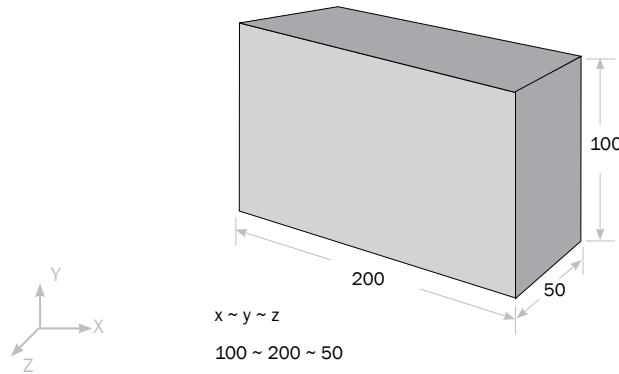
- **Refining surface topology** (<http://altair-2.wistia.com/medias/2p48j67uav>)



12 3D Meshing with HyperMesh

This chapter includes material from the book “**Practical Finite Element Analysis**”. It also has been reviewed and has additional material added by Matthias Goelke.

12.1 When To Use 3D Elements



3D elements should be used when all dimensions are comparable.

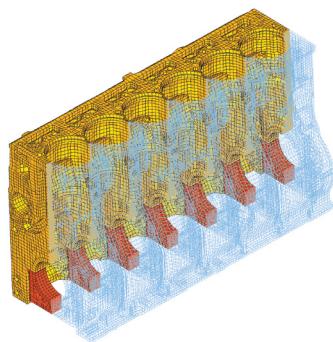
$x \sim y \sim z$

Element shape – Tetra, penta, hex, pyramid

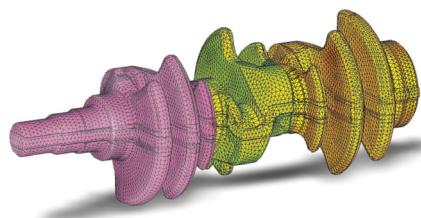
Additional data from user – Nothing

Element type – Solid

Practical applications: Gear box, engine block, crankshaft, etc.



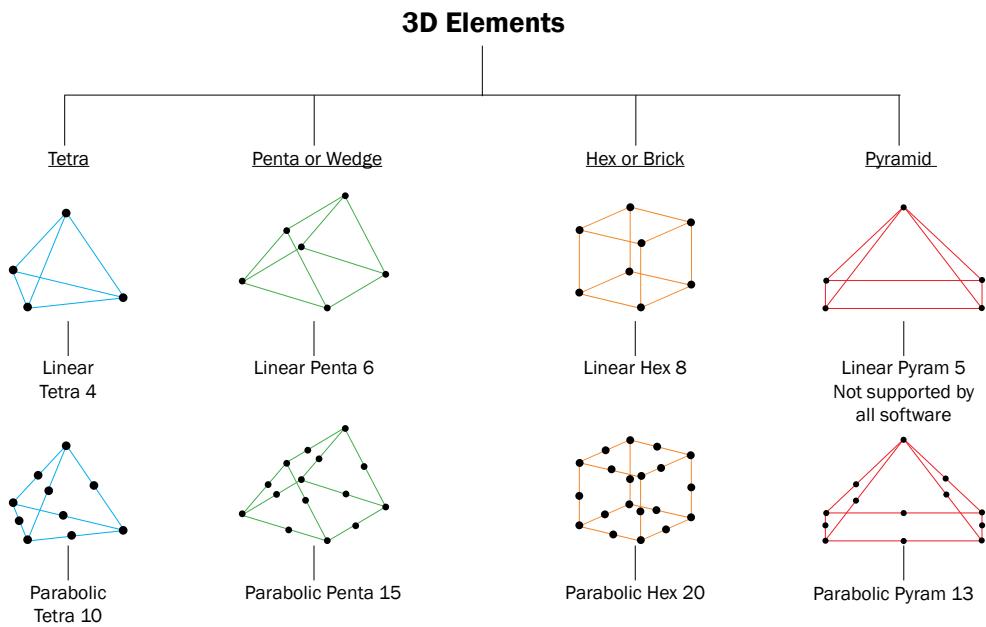
Cylinder crankcase brick mesh



Crank Shaft Tetra Meshing

12.2

3D Element Types



12.3

Dofs For Solid Elements

2D thin shell and 1D beam element supports 6 dofs, but all solid elements have only 3 translational dofs (no rotational dof) i.e. a 10 noded tetra element has total of $10 \times 3 = 30$ dofs

Why does a solid element have only 3 translational and no rotational dofs (physical interpretation)?

Consider a piece of paper (2D geometry) or long steel scale (1D geometry). It could be easily bent and twisted (rotational dof). But now consider a solid object like a duster or a paper weight. It could not be subjected to very high bending or torsion stiffness. Hence, solid elements have been formulated with 3 translational dofs and no rotational dofs.



12.4

Tetra Meshing Techniques

There are two methods of tetra meshing:

1) Automatic mesh: This approach is limited to simple geometries and the pre-requisite is an error free CAD model. The user just has to select the volume and the software automatically carries out the meshing as per the specified element length, quality criteria, etc.

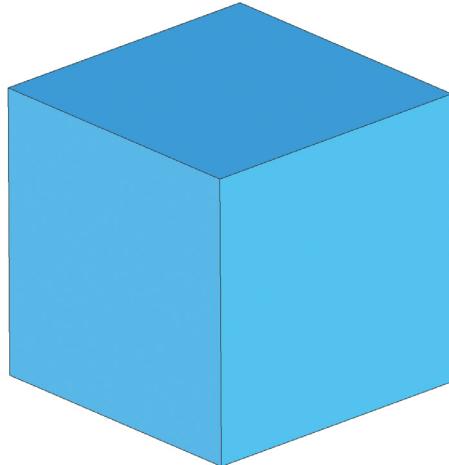
Advantage: Very quick, no meshing efforts

Disadvantage: Results in a very high number of nodes and elements. There is no control over the mesh flow and the specific mesh pattern requirement (like bolted, welded joints or contact surface simulation).

2) 2D (tria) to 3D (tetra): This is the most commonly used method. Quad or tria meshing is carried out on all the outer surfaces of the geometry. During the tetra meshing the quads are automatically split into trias which then serve as the “basis” of the tetra elements.

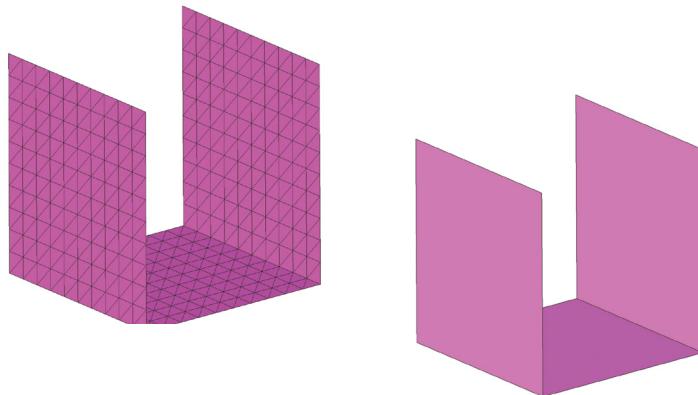
Steps for 2D (tria) to 3D (tetra) mesh generation

Step 1) Study the geometry

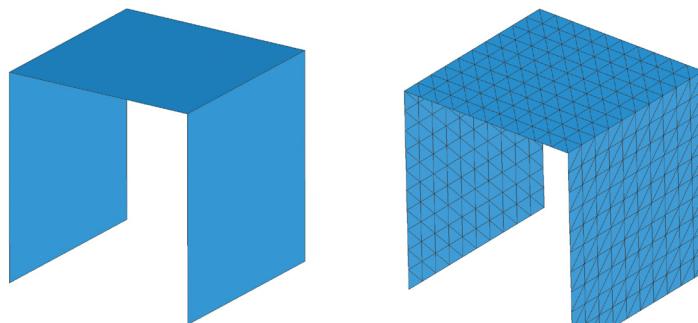


Step 2) Separate (isolate) the surfaces and split the job among engineers (if there is time constraint)

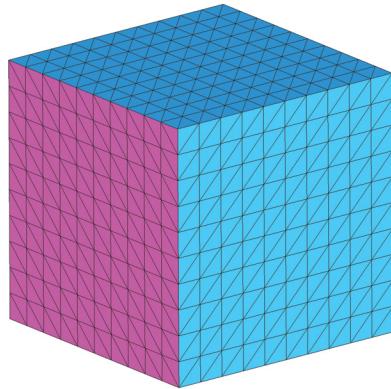
a. CAE engineer 1



b. CAE engineer 2



Step 3) Combine the mesh



Step 4) Perform quality checks for the triangular elements

For instance, min. tria angle > 15°, Max. tria angle < 120°, jacobian > 0.6, zero free edges, no T-connection

Step 5) Convert the tria mesh to a tetra mesh

Step 6) Perform quality checks for the tetra elements (tet collapse > 0.1, Jacobian and distortion > 0.5, stretch > 0.2 etc.). Improve the quality of the mesh if required.

Step 7) Perform a free-free run or otherwise linear static analysis with dummy BC's.

Common Algorithms For Tria To Tetra Conversion

- Advancing Front: This algorithm is very powerful and the most commonly used algorithm.
- Delaunay algorithm
- Tria-quad mesh

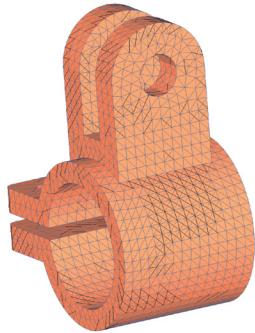
All algorithms provide the following two options for tria to tetra conversion:

1) Floating trias: The original tria mesh generated by the user on the outer surfaces might not match with the software produced tetra. Selection of this algorithm gives freedom to the software to change the triangular mesh pattern (in case of any problem in the tetra mesh generation). This option could be used for the meshing of general components or areas (areas without high stress, components representing stiffness or mass, etc.).

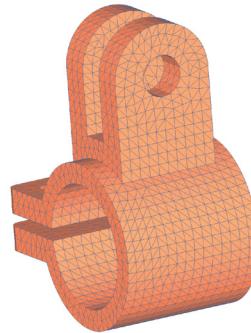
2) Fixed trias: The original tria mesh and the generated tetra mesh pattern match. This meshing option should be used whenever you intend to “glue”/merge individually meshed components together. Otherwise it may happen that the mesh of the components do not match resulting in an incompatibility of the mesh (i.e. the mesh would not be properly connected).



Original tria mesh



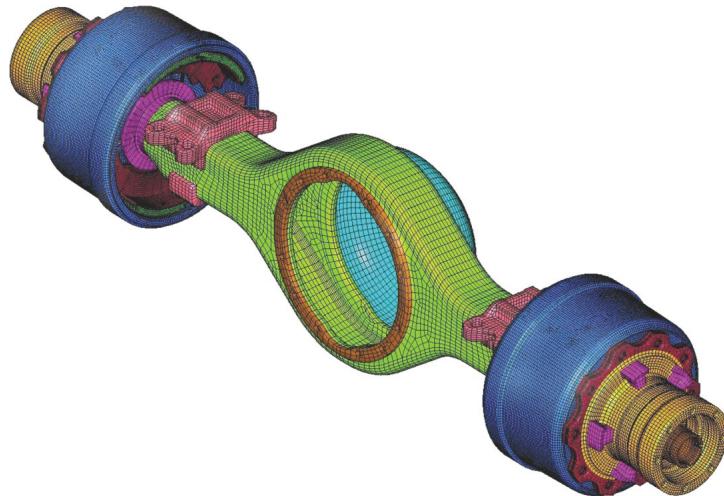
Floating tria method:
Orientation of tria and tetramesh
is not the same (at cross pattern
of mesh)



Fixed tria method:
Orientation of tria and tetramesh
is the same

12.5 Brick Meshing

Brick meshing (also known as hex-meshing) is all about planning, hard work and patience. Brick meshing supports only manual and semi automatic meshing options. Automatic meshing or an option like quad to brick (tria to tetra), is not supported by commercial software.

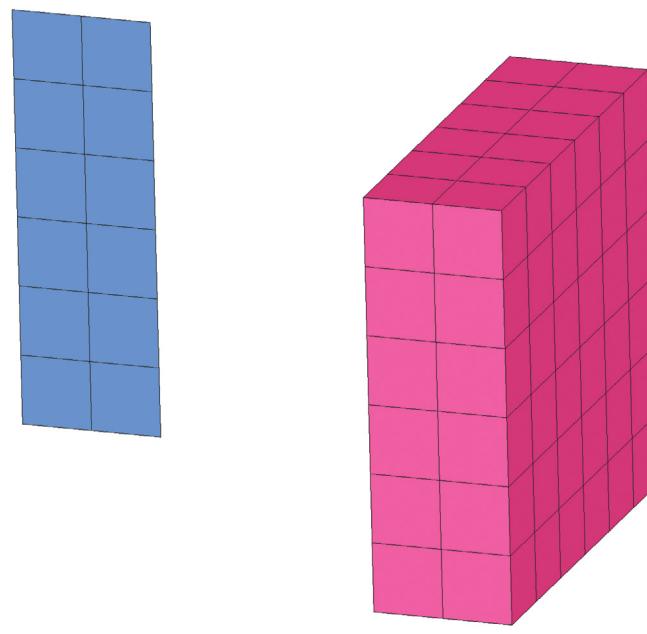


Rear axle assembly with brake drum and wheel hub, brick mesh

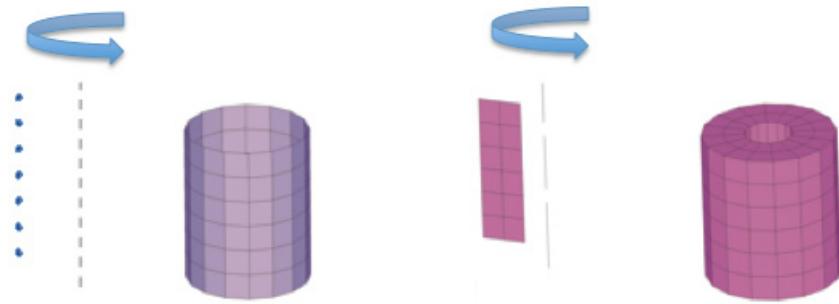
(Image source: Altair Calendar 2006, Courtesy: Ashok Leyland)

The procedure for brick meshing of complicated parts is to first create a shell mesh on the surfaces and then convert this to hex elements by using the meshing commands extrude, spin, sweep, linear solid, etc. Free face check is very important after the completion of hex meshing.

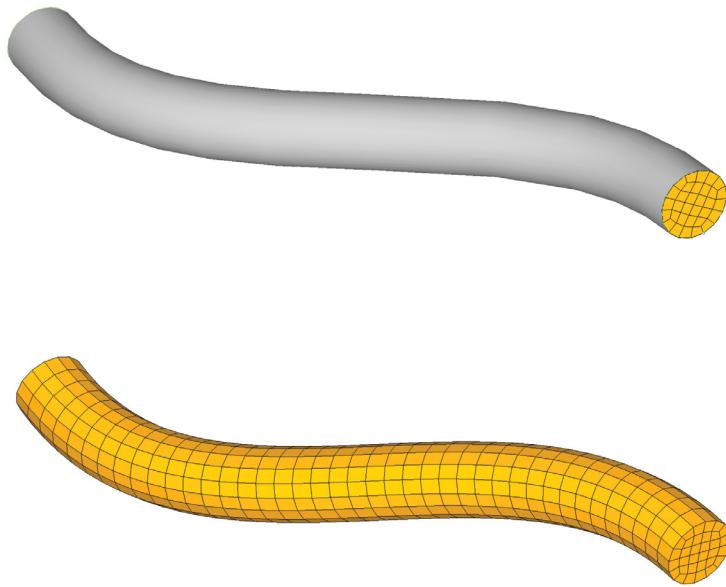
Extrude / Drag



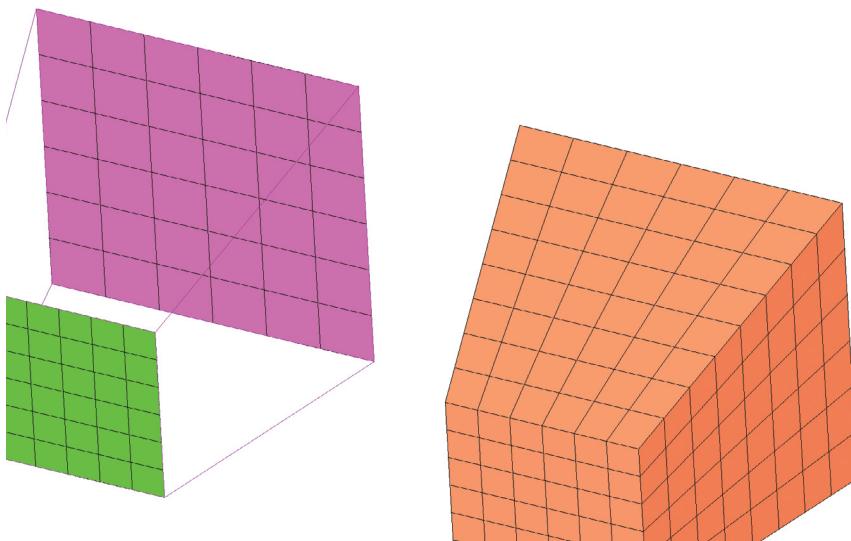
Rotate / Spin



Sweep / Line Drag



Linear Solid



12.6 Tips For Brick Meshing

Even experienced engineers fear and do not willingly accept the job of brick meshing. No book or consultant or university can teach you how to brick mesh. The real teacher is a determined approach and hours of sitting in front of the computer, making mistakes, and learning from the mistakes.

1) Proper planning before starting the job: Sufficient time should be spent in studying the geometry and meshing should not be started until this is done. How to proceed next is visualized in the mesher's mind. Symmetry, sub symmetry or repetitive features are located, and if they exist, this could save you time.

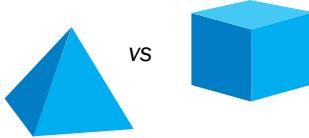
2) 2-D quad mesh should be systematic (ruled or mapped), avoid 2-D auto mesh: Flow lines should be maintained with minimum number of trias and diamond or rotating quads should be avoided. Use of the auto mesher on surfaces sometimes results in a zig-zag or random mesh which might lead to unexpected problems later.

3) Do not hurry to convert the shell mesh to brick: One should not convert a 2D mesh to a 3D mesh immediately. Instead, proceed further with the quads and checking for any possible problem with the current pattern is recommended.

4) Start from the most complicated feature and not the simplest one or a corner of the part: During exams for effective time management, the basic thumb rule told to students is to "Attack the simple problems first and then the complicated one." The thumb rule for brick meshing is exactly the reverse, i.e. "Attack the complicated features first and then the simple ones". Beginners make a common mistake to mesh the simplest and outer corners of the part first.

5) Use the linear solid command: Linear solid and morphing are very powerful commands for brick meshing and should be utilized.

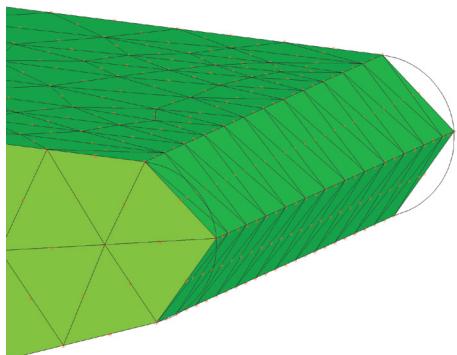
Brick And Tetra Meshing Comparison



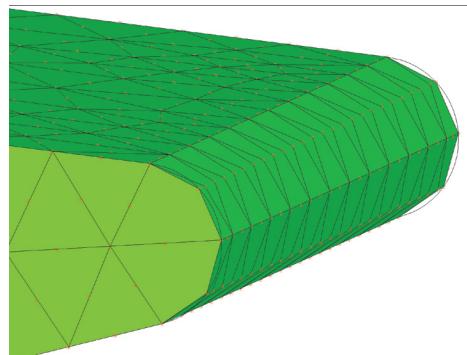
- The number of elements and nodes generated by a brick mesh are of the order of 1/2 to 1/50 in comparison to a tetra mesh. A brick mesh reduces the solution time and results in the ease of handling the model on a workstation (pre and post display).
- Analysis types like crash or nonlinear give preference to brick mesh due to the number of nodes and mesh flow lines.
- The time consumed in brick meshing is more and requires experience, hard work, and a lot of patience too.
- Over the years, the algorithm for tetra meshing has improved and accuracy wise there is not much difference in tetra 10 and brick 8 elements.

12.7 How Not To Mesh

1) Mid nodes should lie exactly on the geometry



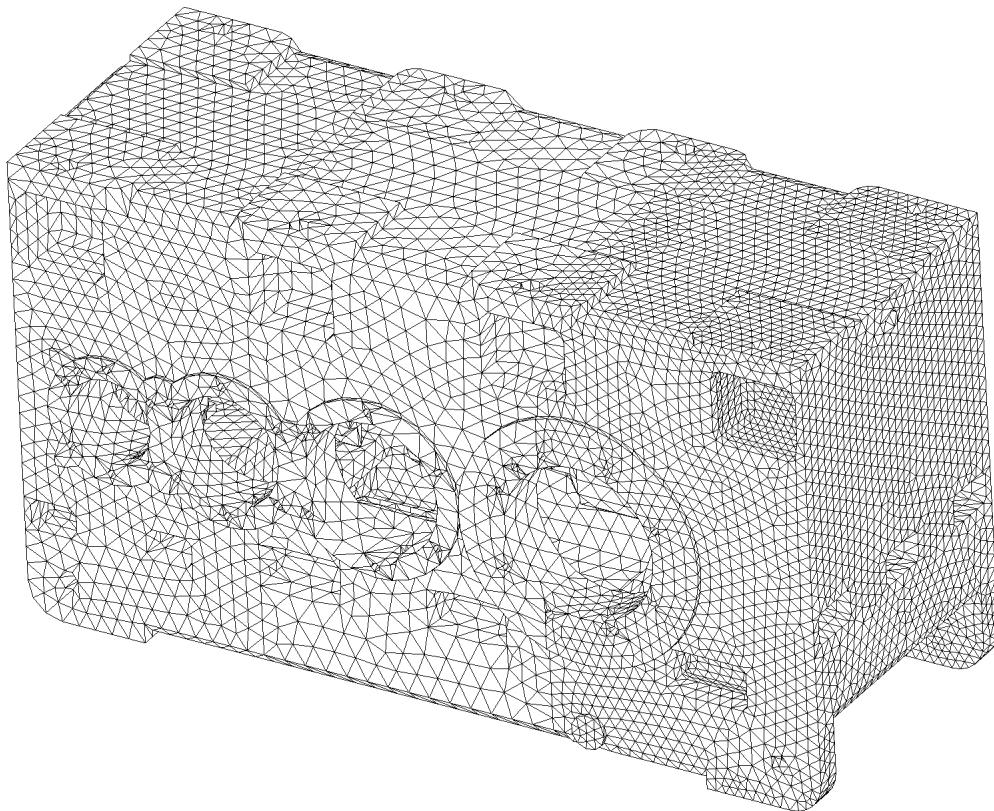
Not acceptable



Recommended

For a parabolic tetra meshing task, many CAE engineers prefer to start with linear tria (instead of parabolic) meshing and then convert it to parabolic. In the conversion process, mid nodes might not get projected automatically on the curved surfaces and fillets. If so, it should be projected on corresponding surfaces before conversion to tetras.

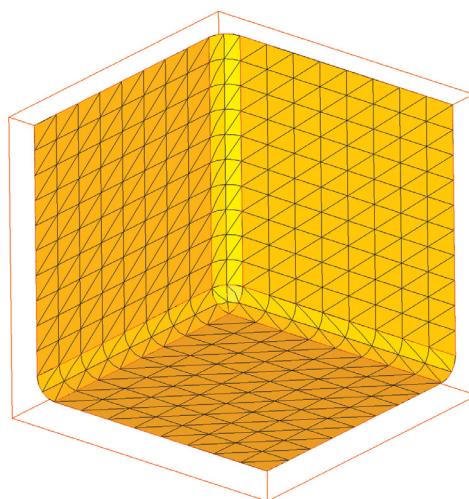
2) When the job is split among several engineers, the element length and over all mesh pattern should be consistent.



The above job was split among 3 engineers due to a very short time duration provided by the client. The same mesh size and pattern was not followed by the engineers working independently on sub parts of the geometry.

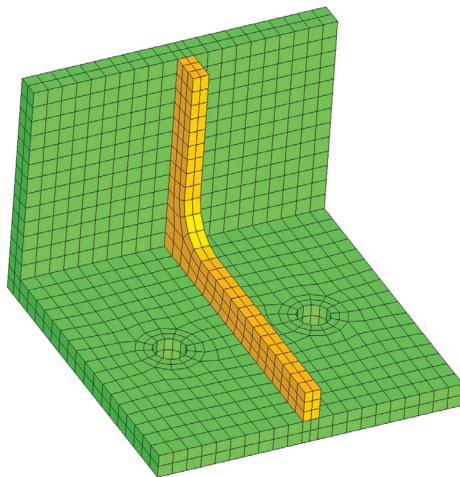
3) Minimum 2 elements on the fillets for tetra meshing

Elements at fillets and curved surfaces usually fail in the jacobian/distortion element check. The manual adjustment for improving the element quality results in mesh deviation from the geometry and visible kinks. This could be avoided by modeling the fillets with 2 or more elements.

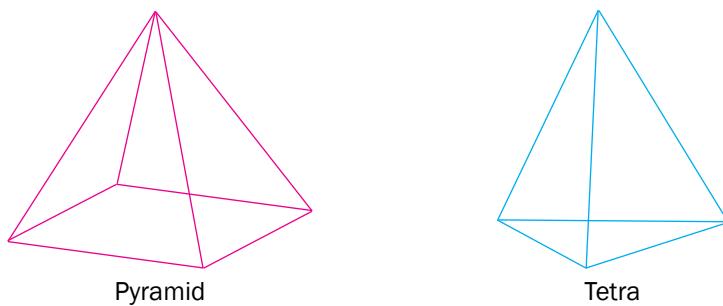


4) For brick meshing, a minimum of 2 elements across the thickness should be used

A single element leads to a poor interpolation and thus affects the accuracy of the results. A minimum of 2 elements across any thickness is recommended. The exception is NVH applications, where stress is not the main criteria, but the representation of the mass and stiffness (with the least dofs) is the main criteria.



5) Use of tetra / pyramid elements while brick meshing

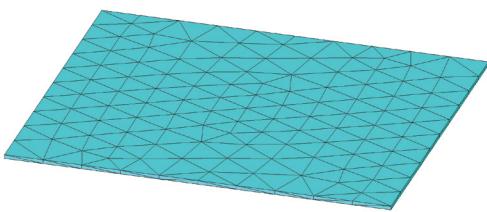


Some clients allow for a few tetra elements during brick meshing. Also some software and analysis types support pyramid elements. Use of tetra and pyramid elements can make the life of a brick mesher tolerable. It's good practice to clarify the instructions for the use of these elements from the client.

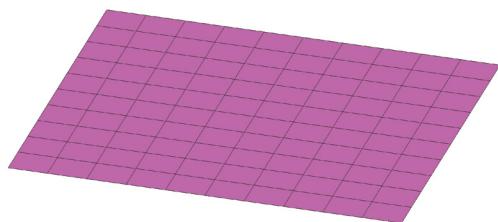
6) Modeling a sheet metal part with 3D elements

For sheet metal or very small thickness parts, 2D shell elements are better suited and recommended. It's not like we cannot use a 3D mesh, but it will result in a very high number of nodes and elements.

Consider the following sheet metal part (200 x 200 x 2 mm). We will mesh the same part with 3D parabolic tetra elements and 2D quad-4 (linear) elements using the same element length and compare the number of nodes and elements needed.



3D Tetramesh
Nodes = 1496 Elements = 689

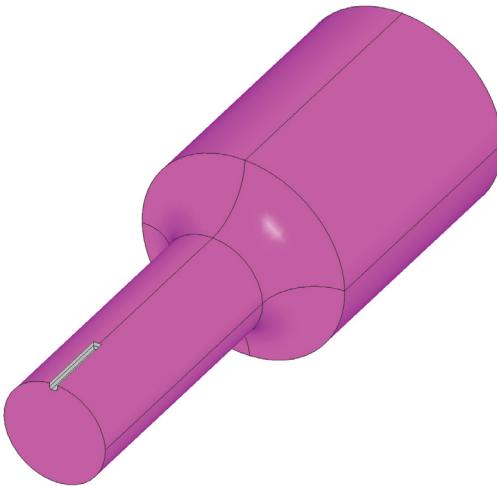


2D quad-4 mesh
Nodes = 121 Elements = 100

7) Limitation of 1D element and advantage of 3D meshing

Fillets, cutouts and complicated geometry features cannot be represented accurately by 1D elements.

3D elements, because of 3 dimensions, can capture all the minute details accurately. For example, consider the following shaft. It is very difficult to capture the key way slot and variable fillet using 1D elements. Instead 3D meshing is recommended for such applications.

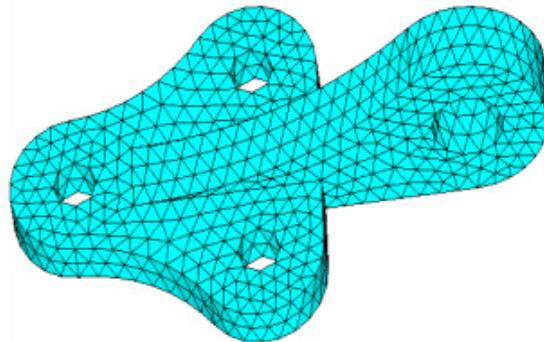
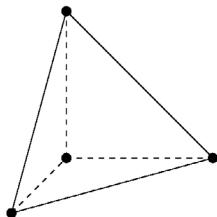


12.8 Creating 3D Elements Using HyperMesh

HyperMesh can be used to generate a tetra mesh or a brick mesh. Below is an explanation on how to perform both types of meshing.

Tetra Elements

Tetrahedral (tetra) elements are solid elements which have been extracted from 2D tria elements. They have a tetrahedral shape as shown below:



HyperMesh builds 4- and 10-noded tetras. Tetra elements can be created through the edit element panel, tetra mesh panel, Tetramesh Process Manager template, and shrink wrap panel.

Edit Element Panel

Tetra elements can be created manually by using the edit element panel. The “Edit Element” panel allows you to hand build, combine, split, or modify elements. Tetra elements can only be created in the edit element panel through the create subpanel.

Tetramesh Panel

Tetra elements can be automatically created using the tetramesh panel. The Tetramesh panel allows you to fill an enclosed volume with first or second order tetrahedral elements. A region is considered enclosed if it is entirely bounded by a shell mesh (tria and/or quad elements).

Different sub-panels exist for different types of tetra meshing:

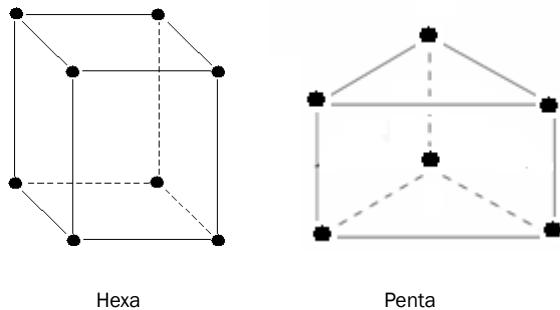
- tetra mesh: allows you to fill an arbitrary volume, defined by its surface using tria/quad elements, with tetrahedral elements
- tetra remesh: regenerates the mesh for a single volume of tetrahedral elements
- CFD mesh: allows you to automatically generate meshes with boundary layer type elements (pyramids and hexas) from selected boundary regions/elements, and fill the remaining core volume with tetrahedral elements
- volume tetra: given a solid entity or a set of surfaces representing a closed volume, this meshing option generates a shell mesh and fills the enclosed volume with solid elements

Shrink Wrap Panel

Within the shrink wrap panel there is an option to generate solid mesh – this will produce an all-hexa or all-tetra mesh based on the selected elements or geometry. The shrink wrap can thus be used as a quick mechanism to generate solid meshes. Note that when generating such a mesh, the Jacobian value has a large effect on the coarseness of the resulting volume mesh. For additional information, please refer to the online help.

Hexa And Penta Elements

Hexahedral (hexa or brick) elements are solid elements which have been extracted from 2D quad elements. Pentahedral (penta) elements are solid elements which have been extracted from 2D tria elements. Their shapes are shown below:



HyperMesh builds 6- and 15-noded pentas and 8- and 20-noded hexa elements. Pentas and hexas can be built in any of the following panels:

- drag: Drags a group of two-dimensional elements along a vector to create solids.
- edit element: builds elements by hand
- line drag: Drags a group of two-dimensional elements along a line
- linear solid: creates solid elements between two-dimensional elements

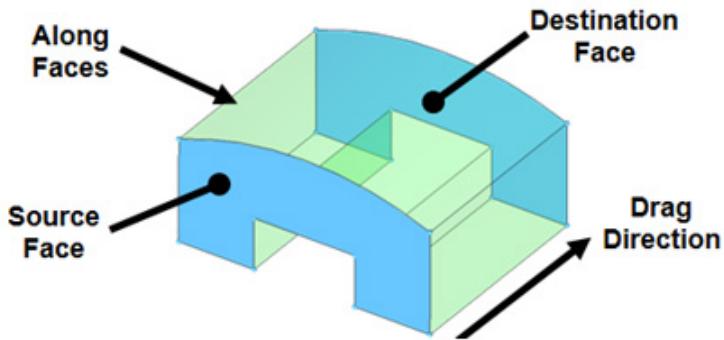
- [solid map](#): builds solid elements between nodes, lines and surfaces
- [solid mesh](#): builds solid elements between a variable number of lines
- [elem offset](#): creates solid elements by offsetting a group of two-dimensional elements normal to the surface formed by the group of two-dimensional elements
- [spin](#): spins a group of two-dimensional elements about a vector to create solid elements
- [split](#): propagates split hexas

As stated above, pentas and hexas can be created in the solid map panel. Below is a detailed description of this panel.

Solid Map Meshing

In the Solid Map panel, solid elements are created. These solid elements can be created from solid geometry as well as from nodes, lines, and surfaces. Below is a short description on mappability. For a detailed explanation on how to split a solid geometry for solid meshing, please refer to the online help topic Partitioning Solids for Mappability.

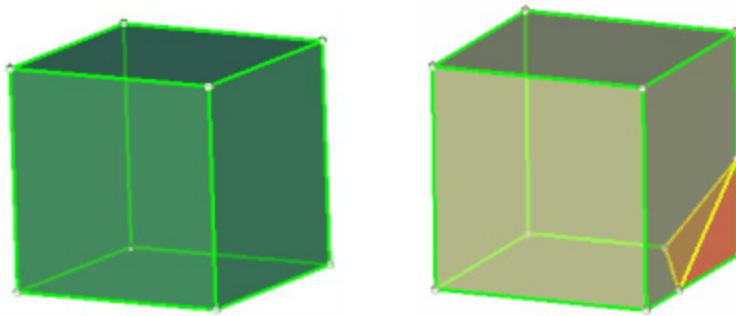
In solid meshing, the ability to be meshed is referred to as mappability. Mappability is directional and can be likened to putting a surface mesh on one face of the solid, then extending that mesh along a vector through the solid volume. So, for example, a perfect cylinder is mappable in one direction (the axis between its top and bottom faces) while a perfect cube is mappable in three (the axes between each pair of its identical faces). However, a combustion engine's cylinder head consisting of two cylinders of different radius joined together into a single solid entity would need to be partitioned to divide the two cylinders. Once partitioned, each cylinder would become mappable in one direction. Below is an example which shows a volume which is mappable in one direction:



While this example shows two faces that are of the same shape and directly oppose each other, that is not a requirement. The source and destination can be of drastically different shape and contour and need not lie directly opposite each other.

Any given volume can have one of four states, which are color-coded when using the mappable view option on the visualization toolbar. Although the colors can be customized, the default settings are:

- **Blue** indicates a solid that has not been edited at all and therefore isn't evaluated for mappability.
- **Orange** indicates a solid that has been edited, but remains completely unmappable (further partitioning may enable mapping).
- **Yellow** indicates a solid that is mappable in 1 direction.
- **Green** indicates a solid that is mappable in three directions (this is very rare).



In the images above, the first cube is mappable in 3 directions. The second image shows that if a corner is split off, it becomes mappable in only 1 direction and the corner is not mappable without further partitioning.

The “Solid Map” panel is used for solid-map meshing, and this panel includes several sub-panels. The general, line drag, linear solid, and ends only sub-panels all draw from the same set of input controls (the more specialized panels simply filter out the controls that do not apply to their mapping techniques). Note that all of these sub-panels depend on an existing 2D mesh, which is then extrapolated into a 3D mesh based on the parameters you input. The one volume and multi solids sub-panels, however, can automatically create 3D mesh directly on solids as long as the solids you select are already mappable.

Solid Map Panel

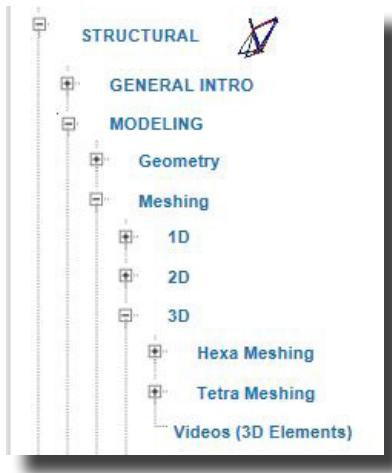
- general: Use the general sub-panel to access all of the possible entry controls for maximum flexibility.
- line drag: Use the line drag sub-panel to select a 2D mesh, and then select a line from the model geometry to use as the mapping direction.
- linear solid: Use the linear solid sub-panel to select two existing 2D meshes and extrapolate a 3D mesh that connects them.
- ends only: Use the ends only sub-panel to select two opposing surfaces and one 2D mesh, then extrapolate the mesh between the surfaces.
- one volume: Use the one volume sub-panel to select a single mappable solid volume and create a new 3D mesh for it.
- multi solids: Use the multi solids sub-panel to select multiple mappable solids and create 3D meshes for them.

12.9 Tutorials And Videos

Recommended Tutorials

The tutorials listed below are included in the HyperWorks installation.

- HM-3200: Tetrameshing
- HM-3210: Creating a Hex-Penta Mesh using Surfaces
- HM-3220: Creating a Hexahedral Mesh using the Solid Map Function



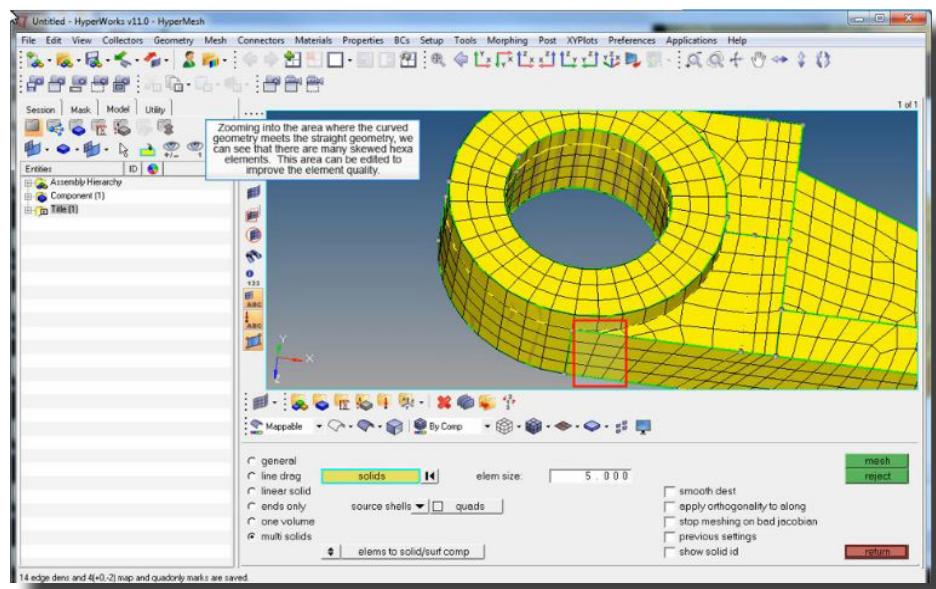
Additional tutorials about tetrahedral and hexahedral meshing are also included in the **Academic Training Center**. In the category Tetrameshing we posted tutorials about tetrameshing of a steering wheel; working with tetra-elements, and local fixing/editing of tetras.

With respect to hexameshing you will be challenged to hexamesh a steering wheel (gives you also an idea about the additional time it takes compared to tetrameshing the same part), and various meshing projects ranging from simple to moderate complicated provided by our team from India.

Recommended Videos

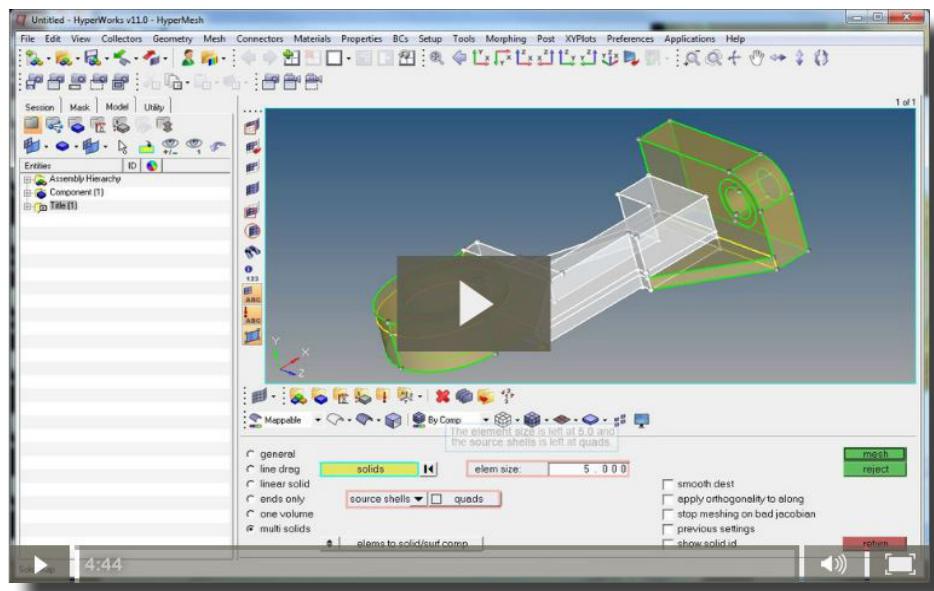
Symmetrical mesh edit

(<http://altair-2.wistia.com/medias/6r7jc6k231>)



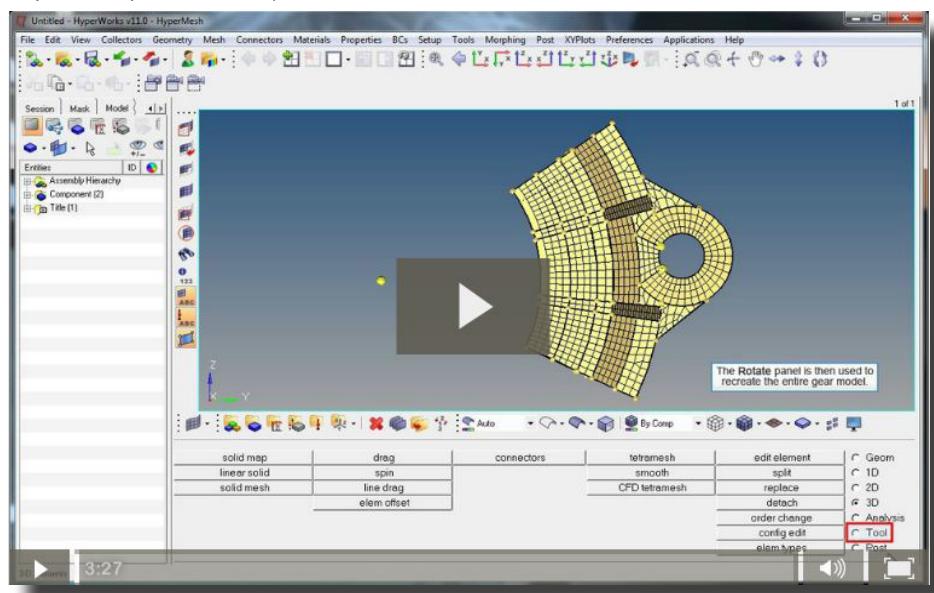
Symmetrical meshing

(<http://altair-2.wistia.com/medias/dbzc76mguv>)



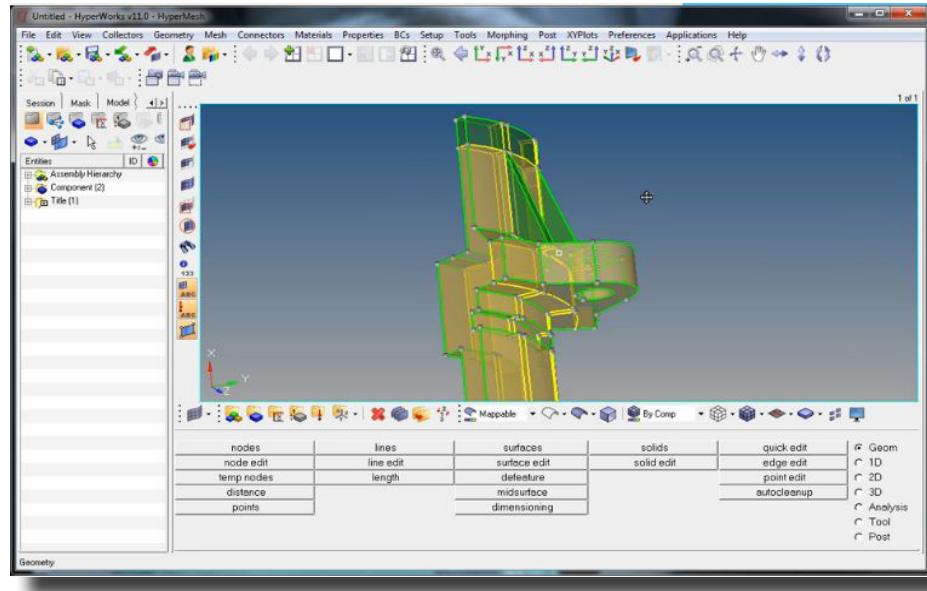
Gear solidmesh

(<http://altair-2.wistia.com/medias/u2lmb0ixfu>)



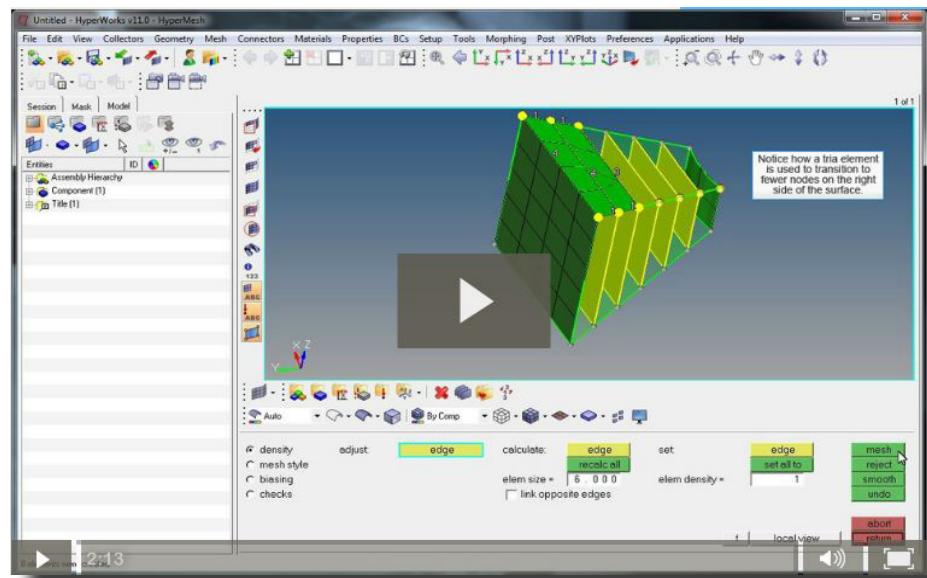
Gear shellmesh

(<http://altair-2.wistia.com/medias/r81811rb7u>)



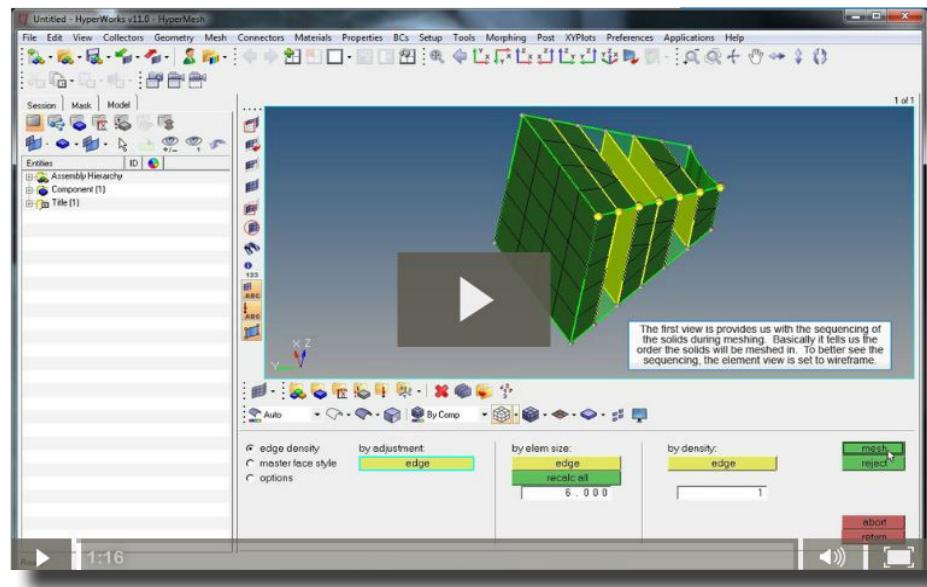
Transition shell

(<http://altair-2.wistia.com/medias/ideze69n5m>)



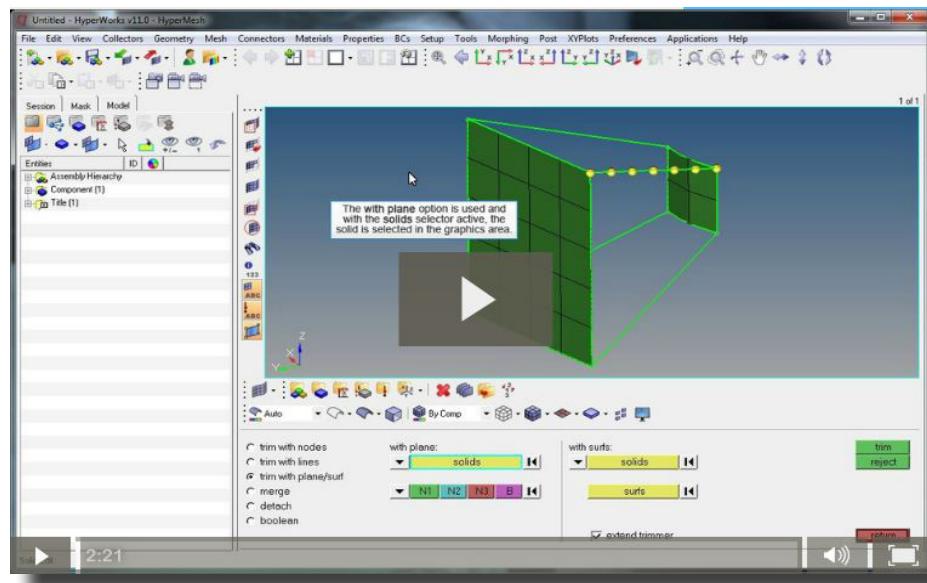
Transition solid

(<http://altair-2.wistia.com/medias/jkm1ht915e>)



Hexa transition

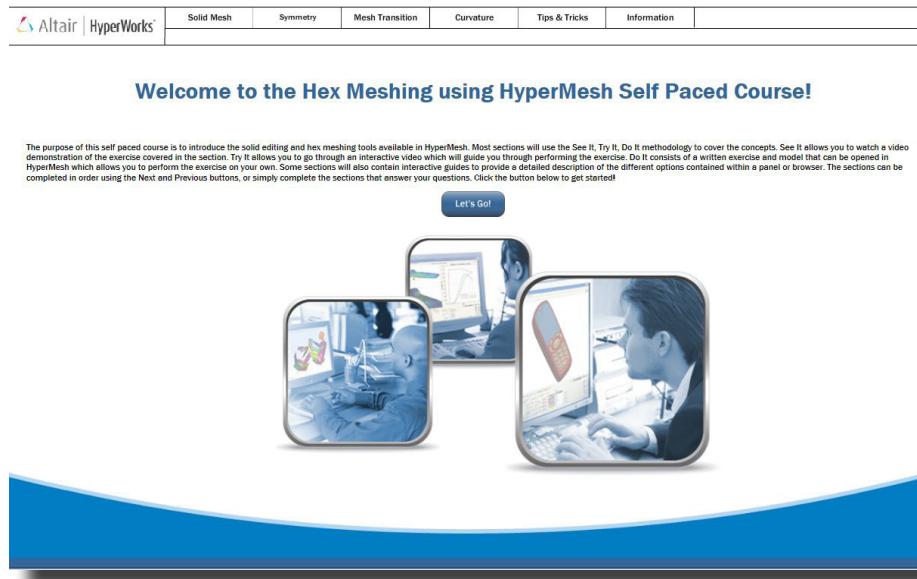
(<http://altair-2.wistia.com/medias/vaclbgnmen>)



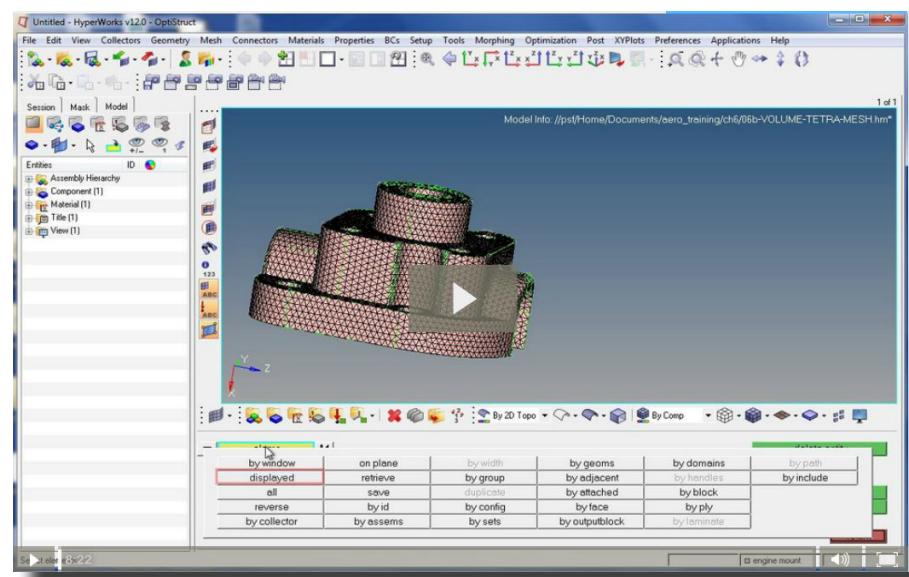
To view the Self Paced Training class about

Hex Meshing using HyperMesh (http://www.altairhyperworks.com/training/self_paced/hex_mesh/index.aspx)

you need to login in the Client Center first (requires an eMail address of your campus)



Tetrameshing (<http://altair-2.wistia.com/medias/fk0y05vnm5>)



Tips and Tricks for Tetrameshing with HyperMesh

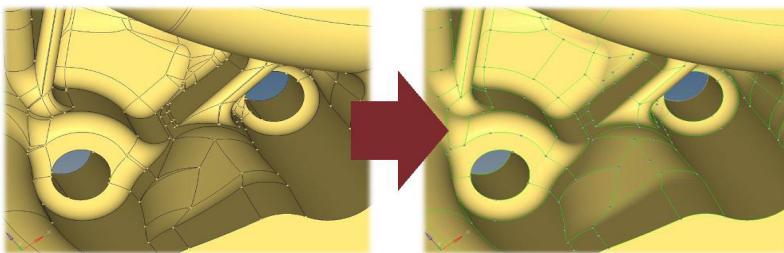
(<https://altair-2.wistia.com/medias/elexrsc3t0>)

Copyright © 2012 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

Autocleanup for Casting Parts

HyperWorks

- Target: suppress double edges and narrow areas, typical of the geometry of casted parts.



Note:

More videos are continuously uploaded to the Academic Training Center: Modeling >Meshing >3D >Videos(3D)

13 Element Quality And Checks

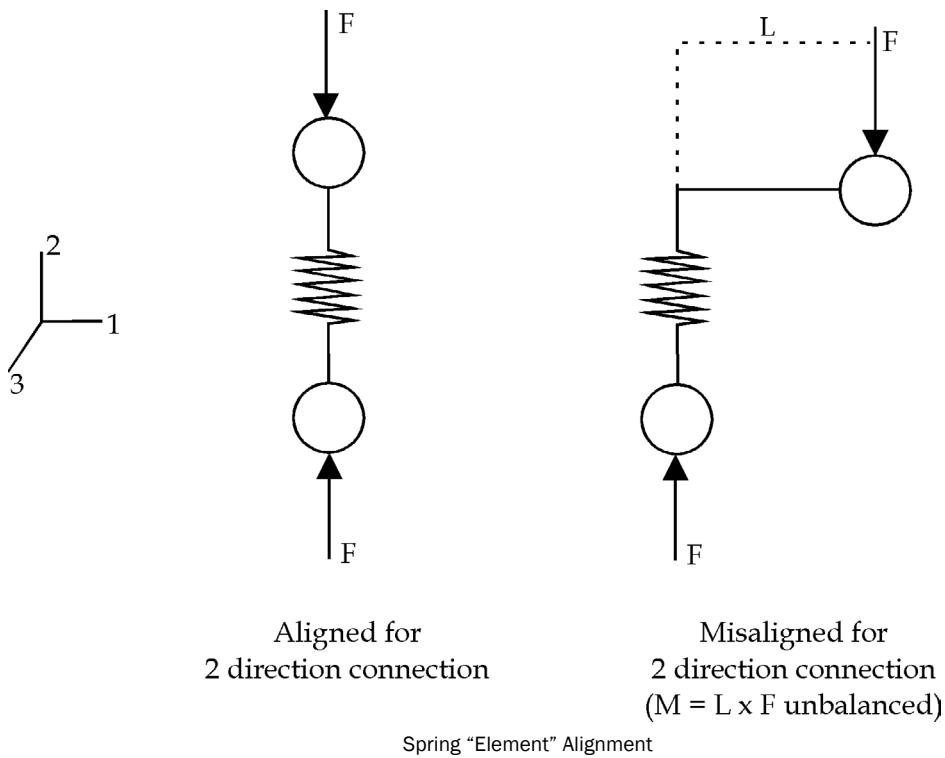
This chapter includes material from the book “**Practical Finite Element Analysis**”. It also has been reviewed and has additional material added by Gareth Lee.

13.1 Compatibility And Mechanisms

Modeling with a variety of element types provides many opportunities to introduce numerical incompatibilities and mechanisms in your structural model. Incompatibilities arise when the element degrees of freedom do not match at a common node, or element faces of solids are mismatched. Mechanisms occur due to some forms of incompatibility or incomplete connectivity. Rigid elements which can model and/or cause a number of interesting problems will be discussed in another section.

Spring Elements

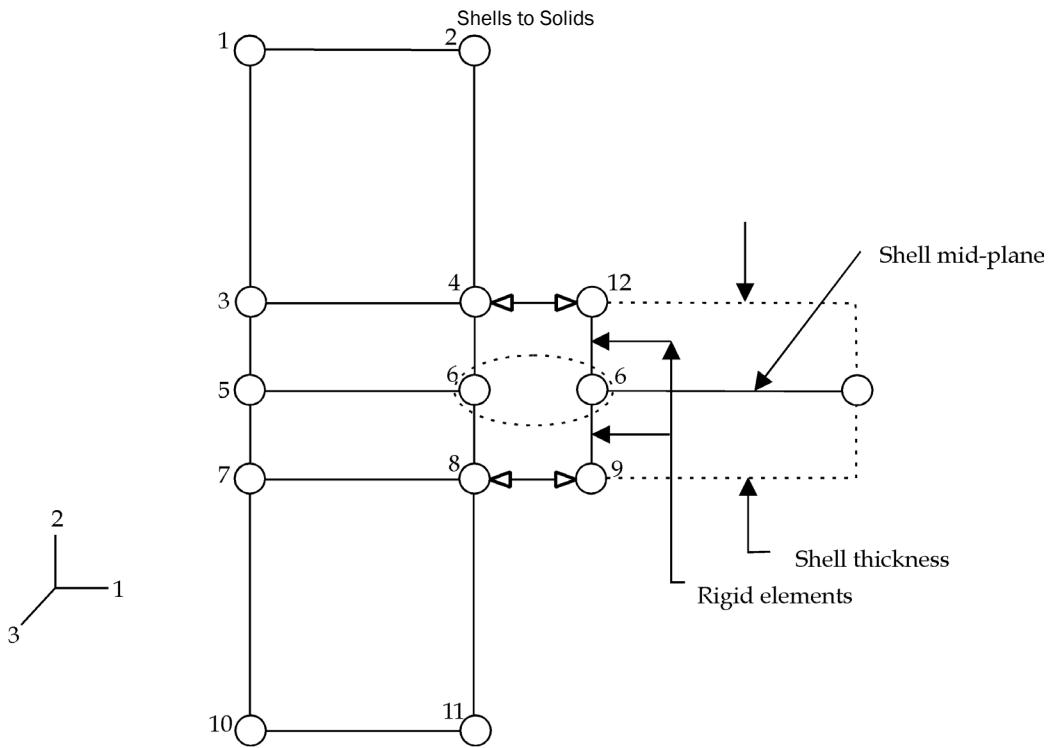
Spring elements are not really elements. The spring is a method of directly coupling two degrees of freedom with a stiffness value or to connect a single DOF to ground through a stiffness. If the DOFs connected are not aligned in the global coordinate system, a mechanism will occur. This mechanism will not cause a singularity but will cause the introduction of a mysterious “torque from nowhere”. The following figure depicts aligned and misaligned springs.



Shells To Solids

The connection of shells to solids is useful in the transition from a compact solid structure to a more shell-like structure. Unfortunately, solids do not support moments applied to nodes and thus more innovative methods are required to connect shells to solids. Running the model with irregular geometry will cause the mesh to lock up and the hinges to be hidden, resulting in answers which appear to be reasonable.

One accurate method of connecting shells to solids is depicted below. This method requires refinement in the solid mesh near the connection to the shell. Also note that local coordinate frames would be required if the normal direction does not line up with the basic coordinate frame.



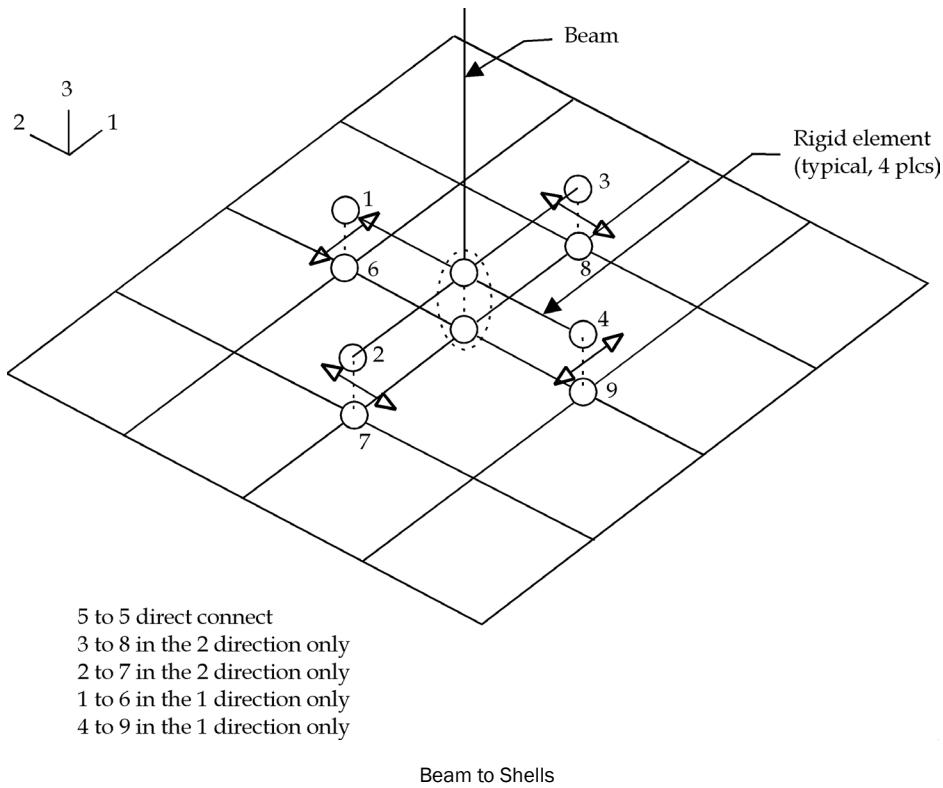
Nodes 4 and 12 are tied in the one direction only.
 Nodes 8 and 9 are tied in the one direction only.
 Nodes 4 and 12, 6 and 8 and 9 are actually coincident but
 have been displaced in this illustration to show their relationship.

Beams To Solids

Beam to solid connections can be handled in a similar manner as shell-to-solid connections. The moments will need to be coupled in at least two directions rather than in only the out-of-plane bending as in the shell-to-solid case.

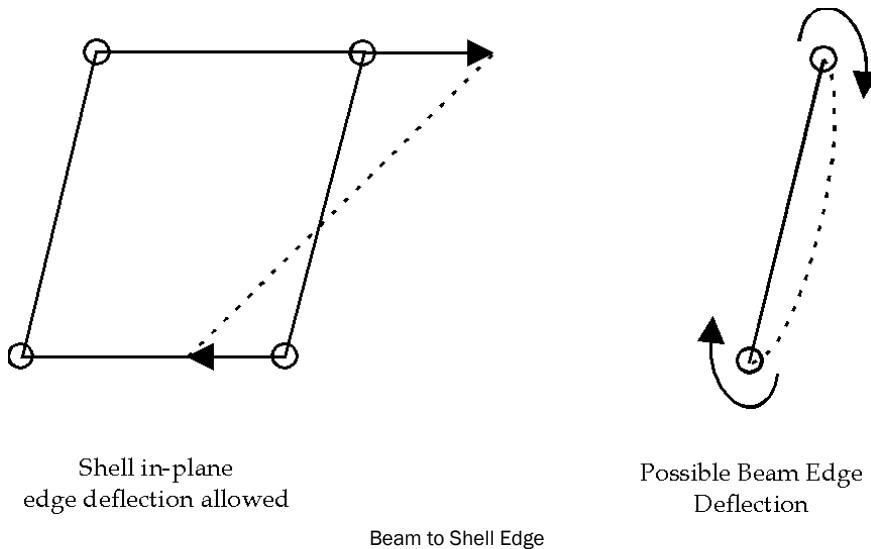
Beams Normal To Shells

Beams connect well to shells except in the drilling direction. This connection can be made by the addition of crank arms to the end of the beam as show below.



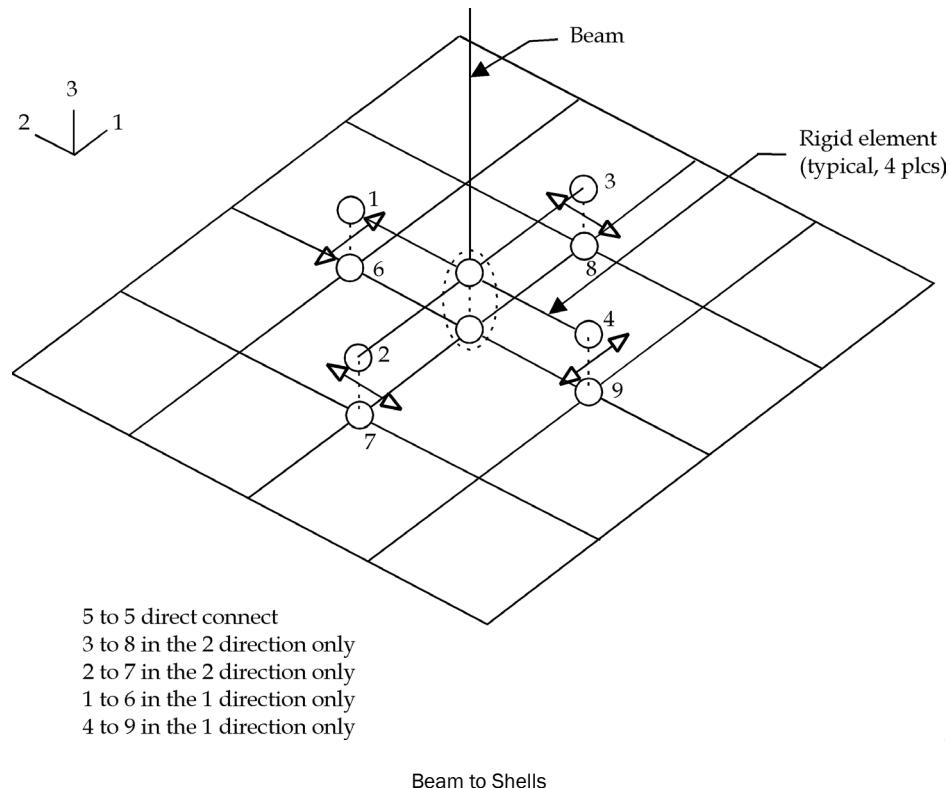
Beam To Shell Edge

An incompatibility exists when connecting beams to shells in the plane of the shell as shown below.



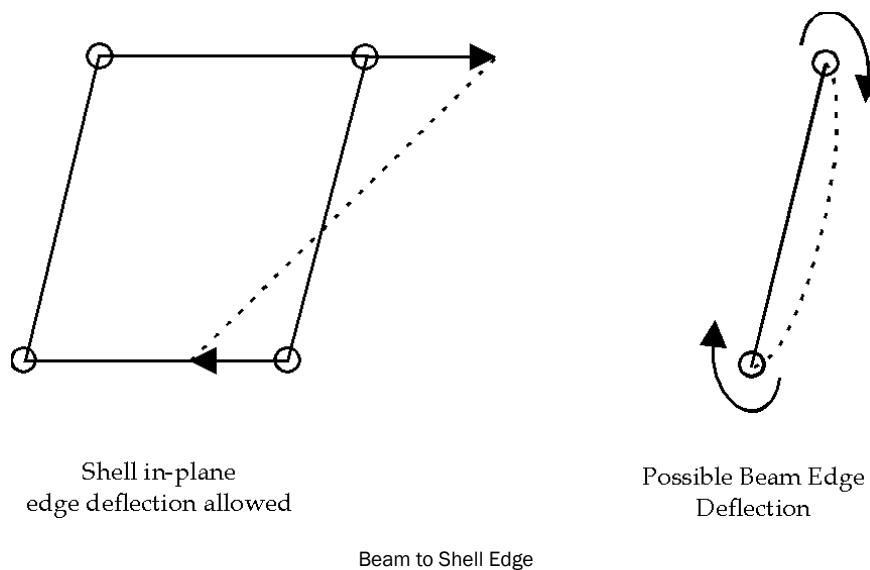
Beams Normal To Shells

Beams connect well to shells except in the drilling direction. This connection can be made by the addition of crank arms to the end of the beam as show below.



Beam To Shell Edge

An incompatibility exists when connecting beams to shells in the plane of the shell as shown below.



13.2 General Element Quality Checks

Element quality is a subject often talked about and never fully understood. The reason for this is complex but is related to the fact that quality is relative and the solution, by definition, is approximate. In the formulation of finite elements a local parametric coordinate system is assumed for each element type and how well the physical coordinate systems, both element and global, match the parametric dictates element quality. Below you see some graphics representing element quality and you should attempt to follow them, however, there will be a point of diminishing return if you try too hard to get every element within the acceptance criteria. Your judgment is your only guide in those cases. Always perform quality checks on the meshes you create. Check with "local experts" regarding the appropriate values for each element type required by your element checking computer programs. Be aware that, in these situations, "correct" answers can vary a great deal as illustrated in the following table where the range between "OK" and "very poor" is quiet wide.

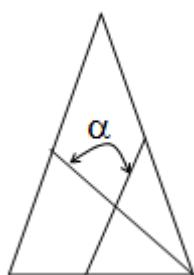
Solid elements use the determinant of the Jacobian Matrix and compare to the ideal value.

Some common element quality measures are detailed below:

BEST	OK	VERY POOR

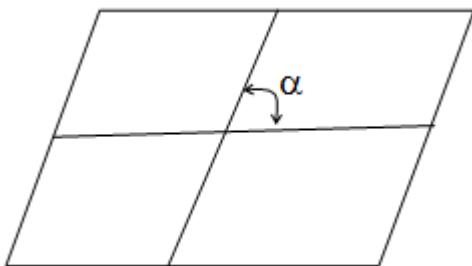
Skewness

Skew in trias is calculated by finding the minimum angle between the vector from each node to the opposing mid-side and the vector between the two adjacent mid-sides at each node of the element. Ninety degrees minus the minimum angle found is reported.



$$\text{Triangle skew} = 90^\circ - \alpha$$

Skew in quads is calculated by finding the minimum angle between two lines joining opposite mid-sides of the element. Ninety degrees minus the minimum angle found is reported.



$$\text{Square skew} = 90 - \alpha$$

The skew check is performed in the same fashion on all faces of three-dimensional elements.

Aspect Ratio

Aspect ratio in two-dimensional elements is calculated by dividing the maximum length side of an element by the minimum length side of the element. The aspect ratio check is performed in the same fashion on all faces of three-dimensional elements.

Warpage

Warpage in two-dimensional elements is calculated by splitting a quad into two trias and finding the angle between the two planes which the trias form. The quad is then split again, this time using the opposite corners and forming the second set of trias. The angle between the two planes which the trias form is then found. The maximum angle found between the planes is the warpage of the element. Warpage in three-dimensional elements is performed in the same fashion on all faces of the element.

Jacobian

The Jacobian ratio is a measure of the deviation of a given element from an ideally shaped element. The Jacobian value ranges from -1.0 to 1.0, where 1.0 represents a perfectly shaped element. The ideal shape for an element depends on the element type. The check is performed by mapping an ideal element in parametric coordinates onto the actual element defined in global coordinates. For example, the coordinates of the corners of an ideal quad element in parametric coordinates are (-1,-1), (1,-1), (1,1), and (-1,1).

The determinant of the Jacobian relates the local stretching of the parametric space required to fit it onto global coordinate space. HyperMesh evaluates the determinant of the Jacobian matrix at each of the element's integration points (also called Gauss points), and reports the ratio between the smallest and the largest.

Different solver codes use different patterns of integration points, and the same solver may use different patterns for different formulations of the same element configuration. One method is as follows:

quad4	(2x2 points)
tria3	(1 point)
tetra4	(1 point)
quad8	(2x2 points)
tria6	(4 point)
tetra10	(4 point)
penta6	(3x2 points)
hexa8	(2x2x2 points)
penta15	(3x2 points)
hexa20	(2x2x2 points)

If the local stretching is the same at all of its Gauss points, then the Jacobian value equals 1.0. As the element becomes more distorted, the Jacobian value approaches zero. A Jacobian value of less than zero represents a concave element, which most analysis codes do not allow.

Both first and second order solid and plate elements are included in this check; however, first order tria and tetra elements always have a Jacobian value of 1.0.

13.3 2D Quality Checks

Ideal shape for quad elements – Square 

Ideal shape for triangular elements – Equilateral triangle 

Different quality parameters like skew, aspect ratio, included angles, jacobian, stretch, etc. are the measures of how far a given element deviates from the ideal shape. A square means that all of the angles are 90° with equal sides, while an equilateral triangle has all angles at 60° with equal sides. Some of the quality checks are based on angles (like skew and included angles), while others on side ratios and area (like aspect and stretch).

To reduce the solution time, elements are mapped to a local coordinate system (individual for every element at the centroid), instead of using a single coordinate system (global). The effectiveness of this transformation is checked by the jacobian and distortion. Ideally all the nodes of a quad element should lie in the same plane, but because of curvatures and complicated geometry profiles, that is not possible. The measure of the out of plane angle is the warp angle.

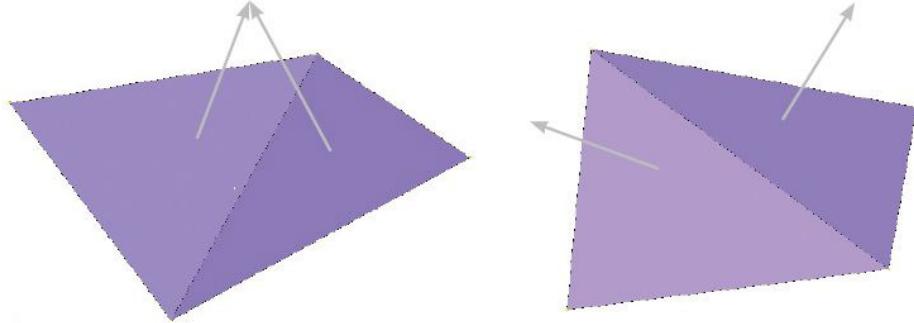
The following are the general definitions of various quality checks. Though the names sound the same, the exact definitions may differ from software to software.

Warp Angle: warp angle is the out of plane angle

Ideal value = 0° (Acceptable $< 10^\circ$).

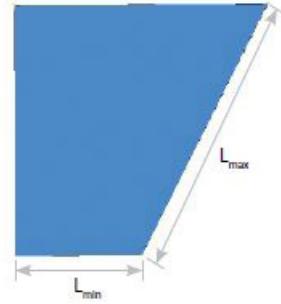
Warp angle is not applicable for triangular elements.

It is defined as the angle between the normals to two planes formed by splitting the quad element along the diagonals. The maximum angle of the two possible angles is reported as the warp angle.



Aspect: maximum element edge length / minimum element edge length

Ideal value = 1 (Acceptable < 5).



Skew

Ideal value = 0 (Acceptable $< 45^\circ$)

Skew for quadrilateral element = 90° minus the minimum angle between the two lines joining the opposite mid-sides of the element (α).



Skew for triangular element = 90° minus the minimum angle between the lines from each node to the opposing mid-side and between the two adjacent mid-sides at each node of the element

Jacobian

Ideal value = 1.0 (Acceptable > 0.6)

In simple terms, the jacobian is a scale factor arising because of the transformation of the coordinate system. Elements are transformed from the global coordinates to local coordinates (defined at the centroid of every element), for faster analysis times.

Distortion

Ideal value = 1.0 (Acceptable > 0.6)

Distortion is defined as - $| \text{jacobian} | * \text{Area}_{\text{LCS}} / \text{Area}_{\text{GCS}}$

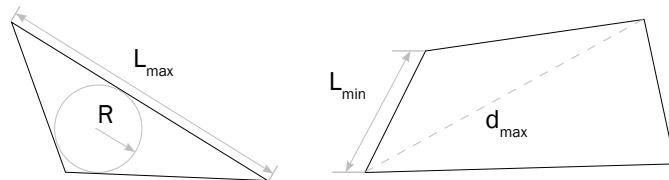
LCS – Local Coordinate system

GCS – Global Coordinate system

Stretch

Ideal value: 1.0 (Acceptable > 0.2)

For quadrilateral elements stretch = $L_{\min} * \sqrt{2} / d_{\max}$



Stretch for triangular element = $R * \sqrt{12} / L_{\max}$

Included Angles

Skew is based on the overall shape of the element and it does not take into account the individual angles of a quadrilateral or triangular element. Included or interior angle check is applied for individual angles.

Quad Ideal value = 90° . (acceptable = $45^\circ < \theta < 135^\circ$)

Tria: Ideal value = 60° (acceptable = $20^\circ < \theta < 120^\circ$)

Taper

Ideal value = 0 (acceptable < 0.5)

$$\text{Taper} = \sum |(A_i - A_{\text{avg}}) / A_{\text{avg}}|$$

$$A_{\text{avg}} = (A_1 + A_2 + A_3 + A_4) / 4$$



Minimum Element Length

This is a very important check for crash analysis (time step calculations). It is also applied in general to check for the minimum feature length captured and the presence of any zero length element.

Chord Deviation

This helps in determining how well curvatures have been modeled. It is defined as the distance between the mid node of an element edge to the curved surface. It is only applicable for linear elements.

How To Improve The Quality Of Poor Elements?

1) Manual adjustment: This is done by translating the nodes manually or remeshing in the poor mesh region. This method consumes lot of time and was the only technique available for years.

2) Drag node: The user has to drag the nodes of the failing elements. It works faster and the advantage is that it instantaneously shows the effect of dragging the node on all the attached elements.

3) Auto quality improvement programs: This is the latest option for quality improvement. The user has to submit the mesh for quality improvement and a software program runs in the background to improve the elements quality automatically.

There is a word of caution for the use of auto mesh improvement programs. For warpage improvement of 2D meshing and Jacobian/distortion improvement of 10 noded tetra meshes, sometimes the software moves the nodes off of the geometry by a considerable amount. This could cause visible kinks and distortion of the geometry.

Apart from the above standard checks, the mesh model should also be subjected to following the additional checks.

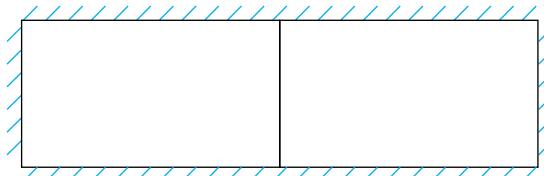
13.4 Other Checks For 2D Meshing

1) Element Free Edges - What is a free edge ?

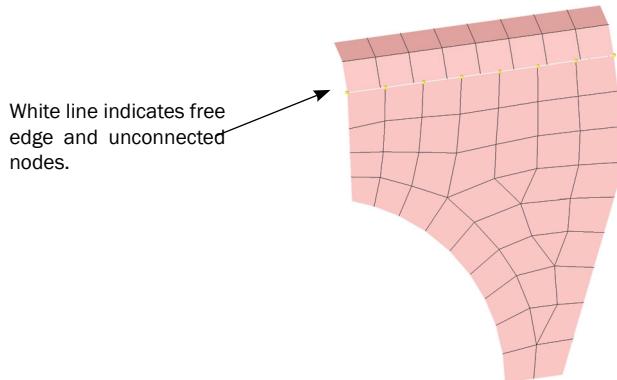
Any single quad element has 4 free edges.



Two elements



In the case above, the middle edge is shared and is no longer a free edge. For a real life FE model, free edges should match with the geometry outer edges / free edges. Any additional free edges are an indication of unconnected nodes.

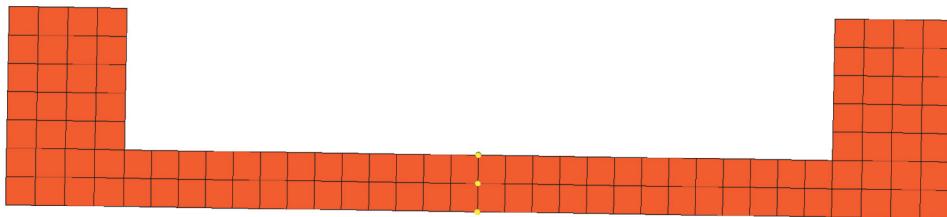


2) Duplicate Element

Mistakes during operations like reflect or translate can result in duplicate elements. These duplicate elements do not cause any error during the analysis but increase the stiffness of the model and results in smaller displacements and stresses. For example, consider a simple plate (thickness = 2 mm) subjected to a tensile load. Assume that due to some meshing operation all the elements are duplicated. If the analysis is carried out then it will show half the stress and displacement.

3) Duplicate Node

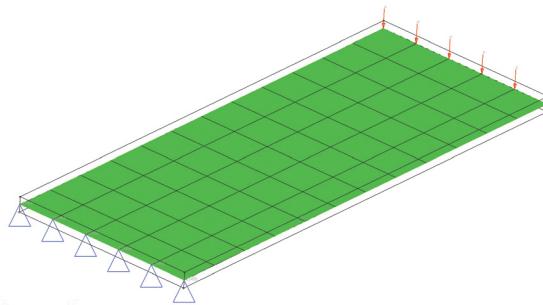
Operations like copy, translate, orient, or reflect can result in duplicate nodes at a common edge.



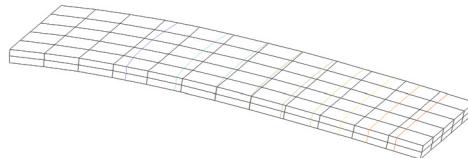
In the above image there are duplicate nodes at the interface where the mesh was reflected. The duplicate nodes are highlighted in yellow.

4) Shell Normal

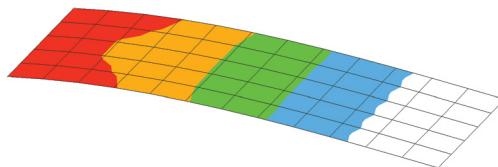
Consider the following example.



After deformation, the top surface of the cantilever beam is under tension and the bottom is under compression.

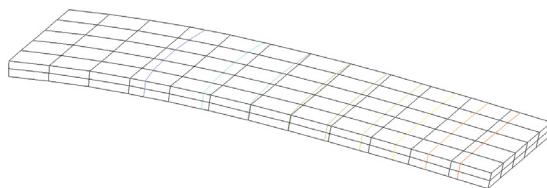


For 2D meshing, a mid surface is extracted and an analysis is carried out on the mid surface mesh. Now the question is, whether the result plot (as shown below) corresponds to the mid surface, the top surface, or the bottom surface?



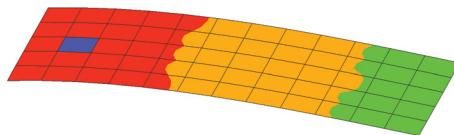
The shell element normal helps us in viewing the top or bottom side stresses. Every element has an elemental (or local) coordinate system. The shell normal is the direction of the element normal (common practice is to represent the normal using the Z axis, assuming the element is oriented in the xy plane). For viewing stress, commercial post processors provide options known as top / bottom or Z1/Z2 indicating the positive and negative direction of the shell normal. The top or bottom is not decided by how the FE model is oriented on the screen but as per the + Z axis orientation of the elements. The Z axis could be displayed on the screen by turning the element triad switch or shell normal vector display option on.

Top side (or Z1) = + Z axis (along the direction of arrow as shown in figure below)
 Bottom side (or Z2) = - Z axis



What Would Happen If The Shell Normals Are Not Aligned Properly?

From an analysis process point of view, there is no error. All of the calculations will be carried out properly. But at the time of post processing when viewing the results for 2D elements, the software does not understand tension or compression. What it recognizes is the shell normal orientation. It can show either the stresses along the + Z axis or -Z axis. Suppose in the above figure, one of the element's shell normal is in the opposite direction. While viewing the results on the bottom side (Z2) all the elements (except the reverse shell normal orientation) show tensile (+) stresses and the odd element shows compressive (-) stresses, as shown in the following figure. The beginner would interpret from the following result that something is wrong with the boundary conditions, but an experienced engineer knows that this is due to an inconsistent shell normal.



How do you correct the shell normal's alignment? The FEA software provides special commands for consistent shell normals (all shell normals aligned in one direction).

5) Geometry Deviation

After the completion of meshing the geometry, the mesh and the geometry should be viewed together (mesh line option off). The mesh should not deviate from the geometry.

6) Delete Free / Temporary Nodes

Free nodes, if not deleted, result in rigid body motion. When the auto singularity option is turned on, the software uses a spring element with a very small stiffness to connect the free nodes with the parent structure. This results in a warning message during the analysis.

7) Renumber Nodes, Elements, Properties Etc. Before Export Operation

Frequent import /export operations could lead to a very large number for the nodes and element IDs. Some software refuse to read the file if the node/element IDs are greater than a specific limit. This could be avoided by renumbering the nodes, elements etc.

8) Observe Type, Family And Number Of Elements (Element Summary For Complete Model)

The mesh should be checked carefully prior to the export operation, as well as after importing it in the external solver for element type, family, numbers, etc. Sometimes due to a translator problem, if properties are not defined properly, or for non supportive elements, either the elements are not exported at all or the family is changed (like membrane elements converted to thin shell, etc.). Plot, trace lines, element free edges, and free faces, if any, should be deleted.

9) Check Mass (actual mass vs. FE model mass)

When a prototype or physical model of the component is available, the FE model mass should be compared with the actual mass. A difference means that there are missing or additional components, or improper material or physical properties.

10) Free-Free Run Or Dummy Linear Static Analysis

Before delivering the final mesh to the client, a free-free run should be performed. 6 rigid modes indicate that all the parts in the assembly are properly connected to each other. In the case of a single component meshing job, a linear static analysis with dummy boundary conditions should be carried out.

11) Request Your Colleague To Check The Model

Due to continually working on the same project, our mind tends to take some of the things for granted and there is a possibility of missing some of the points. It is a good practice to get it cross checked by your colleague prior to final delivery.

13.5 Quality Checks For Tetra Meshes

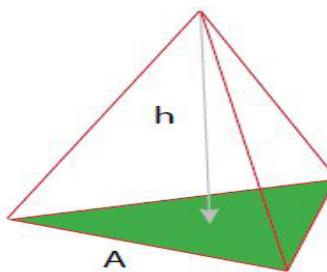
The ideal shape for a tetrahedron element is an equilateral tetrahedron (all equilateral triangle faces). Various quality parameters check how far a given element deviates from the ideal shape.

Tetra Collapse

Ideal Value = 1.0 (Acceptable > 0.1)

Tetra collapse = $h * 1.24 / A$

(Defined as the distance of a node from the opposite face divided by the area of the face multiplied by 1.24)

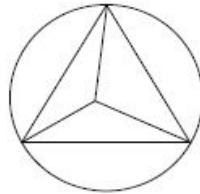


Volumetric Skew

Create a sphere passing through the corner nodes of the tetra, fit an ideal (equilateral) tetra in it. Find the volume of the ideal and actual tetra elements.

Ideal value = 0 (Acceptable < 0.7)

$$\text{Volumetric Skew} = (V_{\text{ideal}} - V_{\text{actual}}) / V_{\text{ideal}}$$

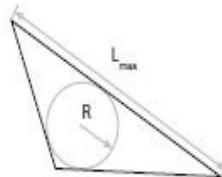


Stretch

Ideal value = 1.0 (Acceptable > 0.2)

$$\text{Stretch} = R * \sqrt{24} / L_{\max}$$

R = Radius of largest possible sphere inside given tetra element.



Distortion

Ideal value = 1.0 (Acceptable > 0.5)

$$\text{Distortion} = |J| * \text{Vol}^m_{\text{LCS}} / \text{Vol}^m_{\text{GCS}}$$

LCS – Local Coordinate System

GCS – Global Coordinate System

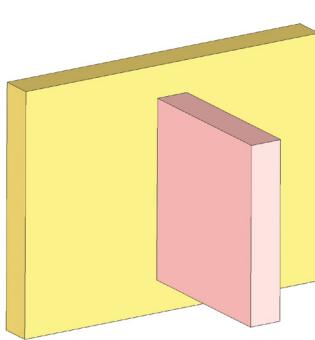
Jacobian

Ideal value = 1.0 (Acceptable > 0.5)

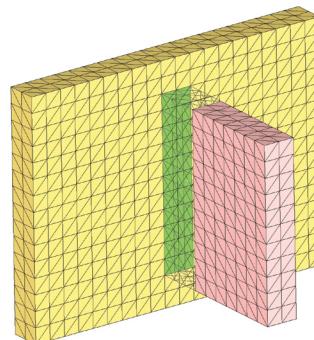
In simple terms, the Jacobian is a scale factor arising because of the transformation of the coordinate system. Elements are transformed from global coordinates to local coordinates to reduce the solution time.

13.6 Other Checks For Tetra Meshes

- 1) Quality checks for 2D tria elements: Before converting trias to tetras, all the quality checks as discussed for shell elements should be applied.
- 2) Free edges: Conversion from tria to tetra is possible only when there are no free edges. No free edges indicate the mesh is enclosing a volume.
- 3) T-connections: The mesh model should not contain any T-connection.

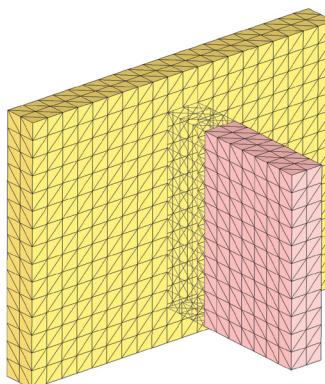


Geometry



T-connection, not acceptable

Green elements should not be there in model



No T-connection (correct mesh)

4) Consistent Shell normals: Before converting trias to tetras, the shell normals should be corrected. Some software do not allow shell to solid conversion unless the normals of all the elements are properly aligned.

5) Geometry deviation: After the completion of meshing, the geometry as well as the mesh should be viewed together (mesh line option off). The mesh should not deviate from the geometry. In the process of quality improvement (in particular for distortion/Jacobian on curved surfaces or fillets), sometimes nodes get translated too far away from the geometry and is not acceptable.

6) 2D tria elements should be deleted before the final submission: It's a common mistake to export 2D shell elements along with tetra mesh in the final delivery.

How To Improve Quality Of Tetra Mesh

Software provides auto algorithms/localized remeshing options for improving the mesh quality. These programs improve most of the elements but some might stay the same. For such elements, the quality improvement is carried out using the manual translation of nodes. Unlike 2D shell elements, 'dragging the nodes' is not an available option for 3-D elements.

Linear vs. Parabolic Tetra Elements

Linear tetra elements are not recommended for structural analysis. They are very stiff and inaccurate compared to parabolic elements.

If Linear Tetra Elements Are Not Recommended For Structural Analysis, Then Why Is This An Option Provided Within Commercial FEA Software?

- 1) For the structural analysis of very big assemblies, to reduce the overall dofs, components away from critical areas are meshed with linear tetras.
- 2) For thermal analysis, a linear tetra is convenient as temperature only has 1 variable (1 dof unlike 3 for structural). The use of tetra 10 elements would unnecessarily increase dofs.
- 3) CFD calculations are mostly based on linear elements.
- 4) Mold flow analysis

13.7 Brick Mesh Quality Checks

The ideal shape for a brick element is a cube. Various quality criteria check how far a given element deviates from the ideal shape.

Warp Angle

Ideal value = 0 (Acceptable < 30°)

Warp angle is calculated on faces (quadrilateral) of a hex element. It is the angle between the planes that form by splitting the quad element.

Jacobian

Ideal value = 1.0 (Acceptable > 0.5)

In simple language, the Jacobian is a scale factor arising because of the transformation of the coordinate system. Elements are transformed from global to local coordinates to reduce the solution time.

Distortion

Ideal value = 1.0 (Acceptable > 0.5)

Distortion = $|J| * \frac{Vol_{LCS}^m}{Vol_{GCS}^m}$

LCS – Local Coordinate System

GCS – Global Coordinate System

Stretch

Ideal value = 1.0 (Acceptable > 0.20); Stretch = min. edge length * $\sqrt{3}$ / max. diagonal length

Aspect Ratio

Ideal value = 1.0 (Acceptable < 5); Aspect ratio = max. edge length / minimum edge length.

Skew

Ideal value = 0° (Acceptable < 45°)

Skew is checked on all the faces of a hex element (quadrilateral). For the skew definition, please refer to the shell element checks.

Quad face included angles: $45^\circ < \theta < 135^\circ$

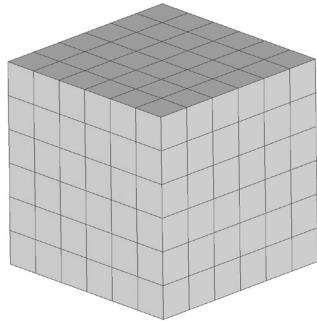
Tria face (wedge / penta elements) included angles: $20^\circ < \theta < 120^\circ$

% of Pentas : Acceptable < 5 %

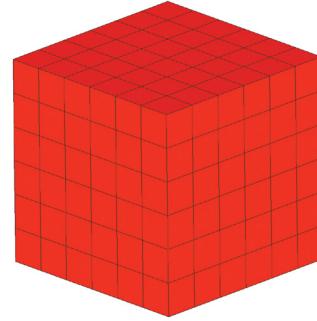
13.8 Other Checks For Brick Meshes

Free Faces

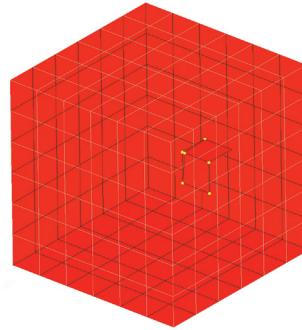
A free face check is the most important check for brick meshing. A single brick element has 6 free faces. Free faces of the mesh should match with the outer surfaces (skin) of the solid part. Any extra (inside) faces indicate that either nodes are not connected properly or there are mismatching elements.



Brick mesh



Free faces

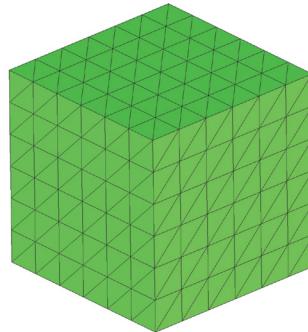


Outer layer elements displayed in line

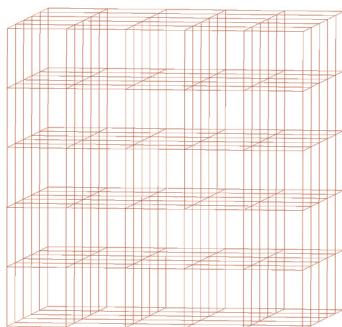
mode to check internal faces

Converting Free Faces To Tetras

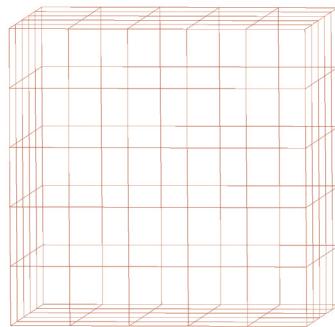
For complicated geometries, checking the internal free faces could consume lot of time. A quick shortcut is to convert the free faces to a tetra mesh. Successful conversion indicates that the brick mesh is OK and there are no internal faces.



Hidden Line Or Dynamic Viewing Option



Line mode



Hidden line mode

By displaying the brick mesh model in the hidden line or fast dynamic viewing mode, one can immediately recognize node / element connectivity problems if there are any.

Apart from the above listed checks, other checks (like duplicate elements, duplicate nodes, deleting temporary or free nodes) as discussed earlier, should also be applied.

13.9 Mesh Check Tools In HyperMesh

HyperMesh supplies many tools for checking the quality of your elements. These tools are located on the 2D page (for checking two dimensional elements), Tool page, and in the Utility Menu on the QA/Model page.

Quality Index Panel

The quality index panel is located on the 2D page and can be accessed by selecting Mesh > Check > Elements > Quality Index. This panel is used to calculate a single value to represent the quality of the displayed two-dimensional (shell) elements. The criteria for the element quality is stored and retrieved using a criteria file. The results from the quality index panel can also be saved to a summary file.

To calculate the quality index, twelve different criteria are used, each with a user-defined weight factor. The twelve different criteria are listed below with their corresponding ideal and worst values.

Feature	Ideal	Worst
min. size	Average element size	0.0
max. length	Average element size	infinity
aspect ratio	1.0	infinity
warpage	0.0	90
skew	0	90
Jacobian	1	-1
max. angle quad	90	180
min. angle quad	90	0
max. angle tria	60	180
min. angle tria	60	0
chordal deviation	0	infinity
% of trias	user-defined	user-defined

Each criterion has five rating levels. Each element is assigned a penalty value according to its rating for individual criteria. Elements failing a criterion are assigned a penalty of 1.0 to 10.0 as a linear function of how far the element is from satisfying the criterion.

Elements passing a criterion have a penalty value of 0.0 to 1.0 for that criterion. The quality index (Q.I.) is a function of individual criteria penalty values. Below is a list of the five levels with a short description. Please refer to the online help for a complete description.

Ideal: This is the absolute best/ideal value that an element can achieve. Elements that fall in this level are drawn in black by default (not highlighted). Ideal elements have no penalty assigned to them.

Good: This level is slightly worse than ideal, but is still considered good for the required analysis. Elements that fall in this level (between good and warn) are drawn in black by default (not highlighted).

Warn: This is an intermediate level between good and fail. This level is used to highlight the elements that have not failed the criteria, but are close to it. The elements that are in this level (that is, falling between warn and fail) are drawn in cyan by default.

Fail: This level determines the elements that are considered to be unacceptable for analysis, thus failed. It is recommended that you fix these elements before performing the analysis. The elements that have failed (between fail and worst) are drawn in yellow by default.

Worst: This level allows you to highlight elements that failed the criteria by a large margin, and which require immediate attention. The elements that fall in and beyond this worst level are drawn in red

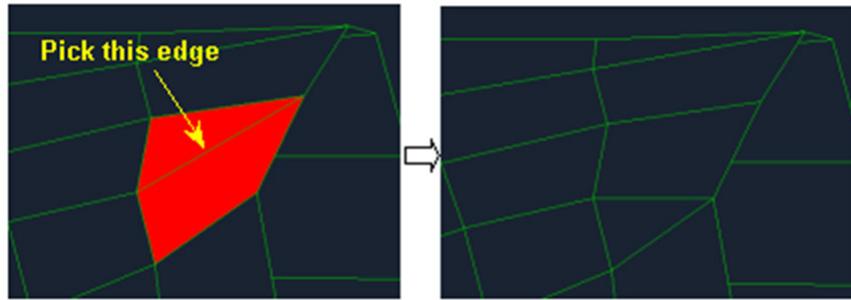
Main Panel Controls

Below is a listing of the controls that are available on the quality index panel. These controls are located on the right-hand side of the panel and include a set of buttons used to optimize the elements for quality check purposes.

display thresholds: This combination of a slider and buttons (one button for each quality level) allows you to visually identify all of the elements at or below a certain quality level. For example, setting the slider to “warn” causes HyperMesh to highlight all elements in the model whose assessed quality level is “warn” or worse—in other words, elements at the “warn”, “fail”, and “worse” quality levels.

place node: While this button is highlighted, click-and-drag a node to interactively reposition it to improve the quality of its attached elements. The nodes only move on the inferred surface and cannot be moved beyond its edges.

swap edge: Change the element connectivity between adjoining elements to improve their quality. When you pick on an element edge, the connectivity of the two elements that share this edge is altered (see image below).



node optimize: Click node optimize and select a node on the screen. HyperMesh repositions the node on the inferred surface to obtain the best possible quality for all elements attached to that node.

element optimize: Click element optimize and select an element on the screen. The locations of all its nodes move on the inferred surface to obtain the best possible quality for that element and its neighbors.

Element Cleanup Panel

The “elem cleanup” panel is also on the 2D page and can be accessed by selecting Mesh > Cleanup Elements > Element Cleanup from the menu bar. It is used to perform automatic cleanup of two-dimensional elements based on the element quality criteria that is set in the quality index panel. You can also specify the element quality criteria by selecting a criteria file. Once the elements have been selected there is an option to use surrounding elements. This option allows HyperMesh to expand your selection to include

additional elements, which provides HyperMesh more freedom to reduce localized distortions. There are also many additional options in this panel for cleaning up the elements. The various options as well as a short description are listed below:

fix folded elements, angle >: Sometimes mesh can get “folded” in such a way that multiple elements occupy nearly the same space. Use this option to combine such elements and simplify the mesh, based on the angle between them. This option requires that an angle be entered. 180 degrees indicates that two elements occupy the same space, so the angle should be close to but less than 180. By default this value is set to 150.

reduce tria elements: This option attempts to combine tria elements into quad elements wherever possible.

keep surf edges: This is an selection under the reduce tria elems option. This setting is used to preserve clean, straight feature lines which were defined from model geometry.

QI smooth elems with target =: When this option is active, HyperMesh attempts to smooth the selected elements until they reach the composite Quality Index value that you specify in this text box.

fix elems failing QI check: This option limits other options by examining whether or not the resulting elements would fail your QI criteria. For example, using tria reduction with this option activated might result in fewer trias being removed, but better overall mesh quality.

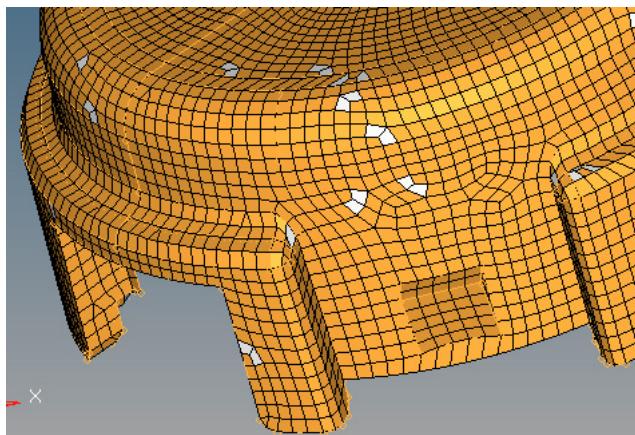
feature angle <: On feature lines with greater angles than this, element cleanup of nodes on the feature lines will only move long the lines. Nodes on feature lines below this angle threshold can be moved in any direction, which may cause jagged features.

Check Elements Panel

The “check elems” panel is located on the Tool page or by selecting Mesh > Check > Elements > Check Elements from the menu bar. This panel is used to verify the basic quality of your elements. There are subpanels for 1D element, 2D element and 3D element checks with the appropriate element quality checks on each subpanel. This panel allows for each element quality check to be performed individually.



For example, on the 2D subpanel, the check for the Jacobian can be performed. This is done by entering the desired Jacobian value to be checked and then clicking on the green Jacobian button. This then highlights all the elements displayed in the model which have a Jacobian value of less than the value specified.

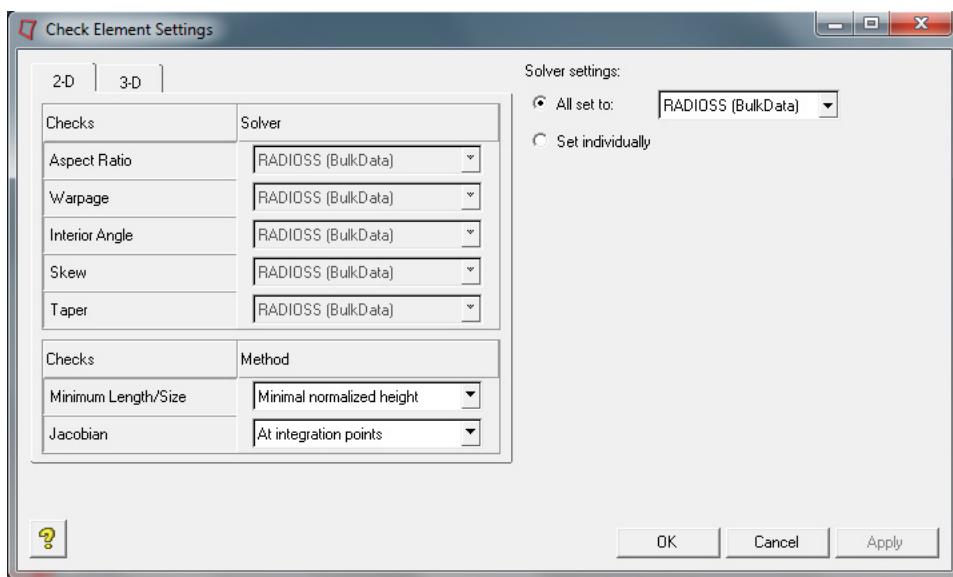


In addition to the element checks on the 1D, 2D, and 3D subpanels, there are also the following functions:

connectivity: This option tests the connectivity of a group of elements

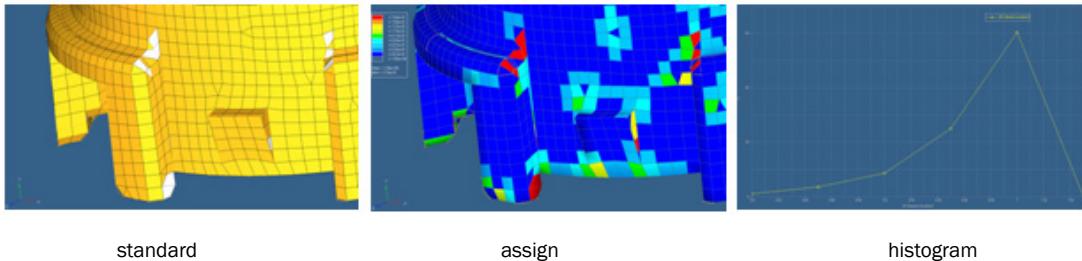
duplicates: This function checks for duplicate elements

settings: This opens the Check Element Settings window. This window allows the user to determine which version of the element check to use. Each check can be set to use a different solver's check method, or the user can set everything to the same method.



save failed: This saves the elements which fail the element check and places them on the user mark. These elements can then be retrieved in a different panel.

standard, assign plot, histogram: This switch is used to determine which view mode should be used.



In addition to the 1D, 2D, and 3D subpanels, there are three additional subpanels available:

time: The time subpanel calculates element time steps and checks for steps that fall below a specified value. To use this function, you must have the template function, HM_CALC_TIMESTEP defined in the current template.

user: This subpanel allows you to specify a template file that checks for any type of user-defined quality

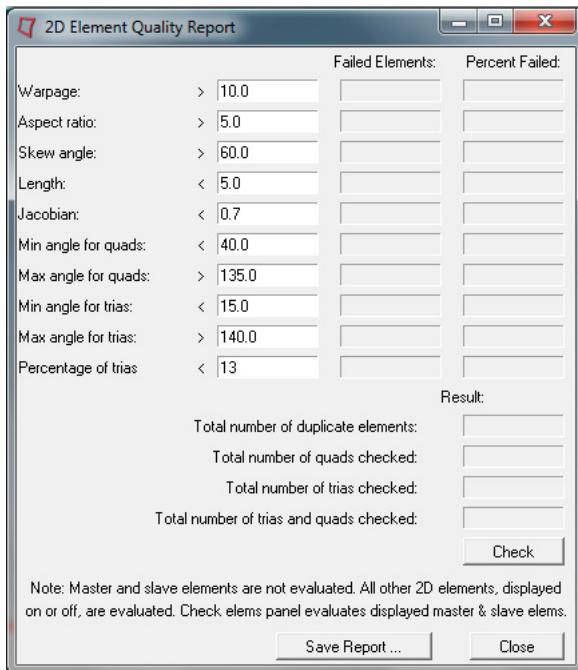
group: Using this subpanel you can delete interface elements that are not attached to a normal element

Utility Menu - QA/Model Page

The QA Utility Menu contains many tools to help you quickly review and clean up the quality of a pre-existing mesh. Under QA Tools, there are multiple macros to check the element quality of the model. The element quality criteria used by these tools comes directly from the values entered on the check elements panel. There are eight tools to isolate elements that fail certain element check criteria: Length, Jacob, Warp, Aspect, Max ang: Q, Min ang: Q, Max ang: T, and Min ang: T. Each of these tools checks all the displayed elements for the selected criteria and highlights those elements which fail.

Another tool available is the Quality Report macro. This macro launches a user interface that allows the user to set the various quality values and check the quality of all the 2D elements in the model.

The results are shown as the number of elements and percentage of elements failing each criterion.



13.10 Mesh Check Tools Tutorials And Videos

Recommended Tutorials

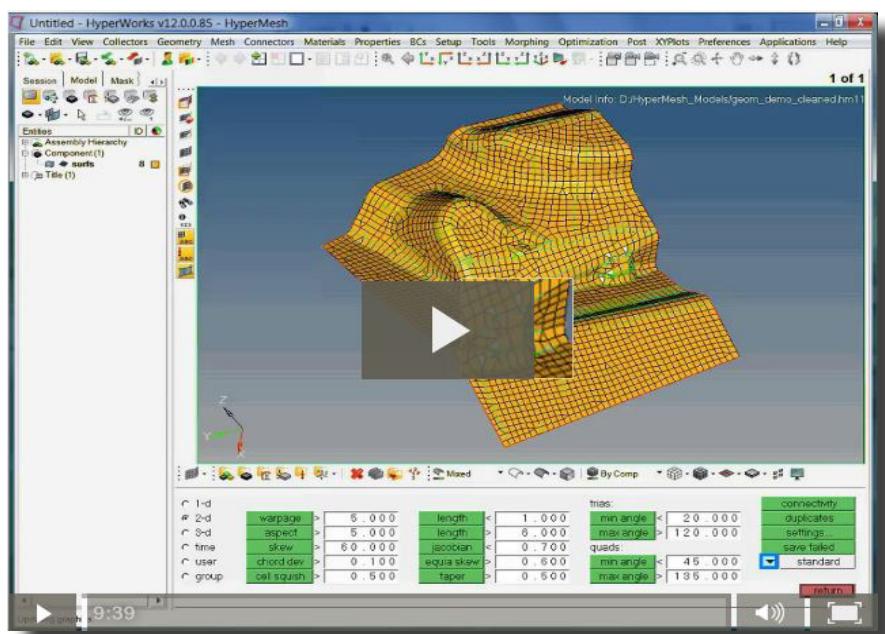
The following tutorials are part of the HyperWorks installation:

- HM-3300: Checking and Editing Mesh
- HM-3320: Penetration

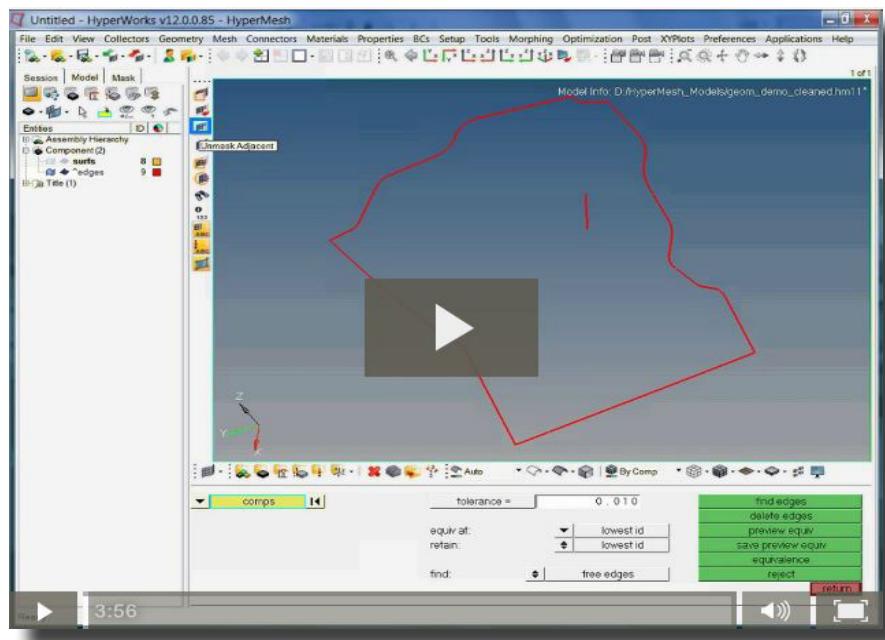
Recommended Videos

You need to be online to view the following videos

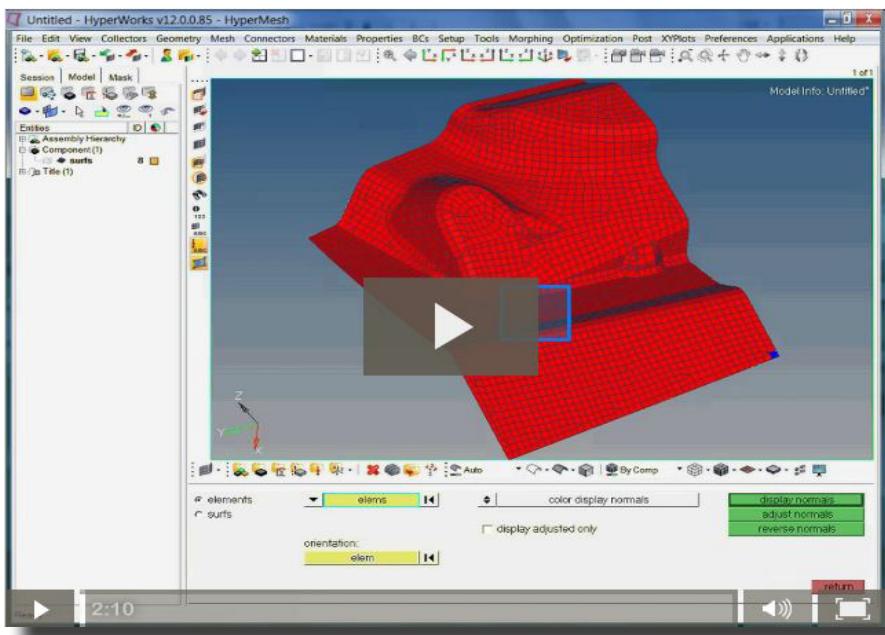
Mesh quality (<http://altair-2.wistia.com/medias/dog2q02070>)



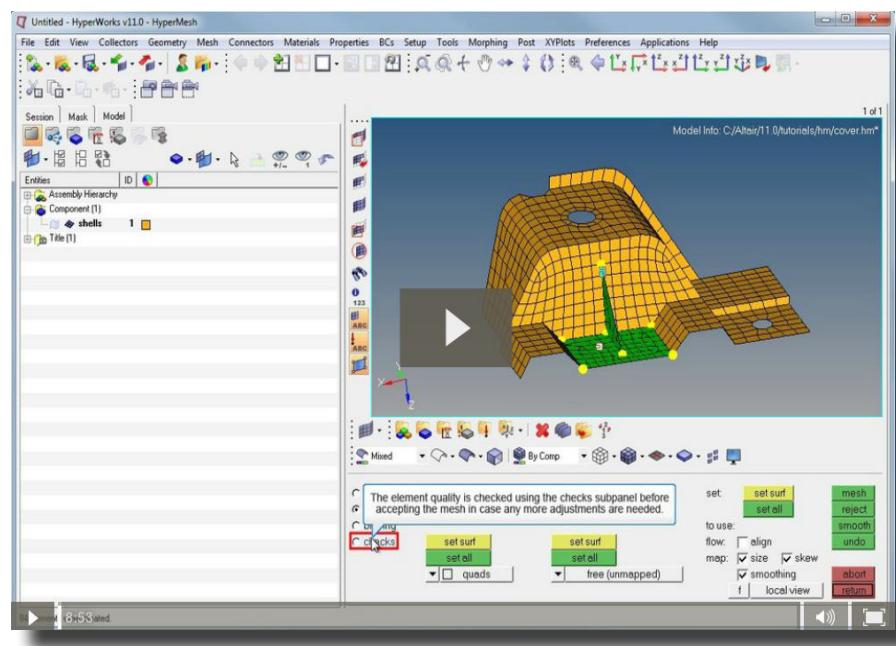
Free element edges (<http://altair-2.wistia.com/medias/4im0absvzu>)



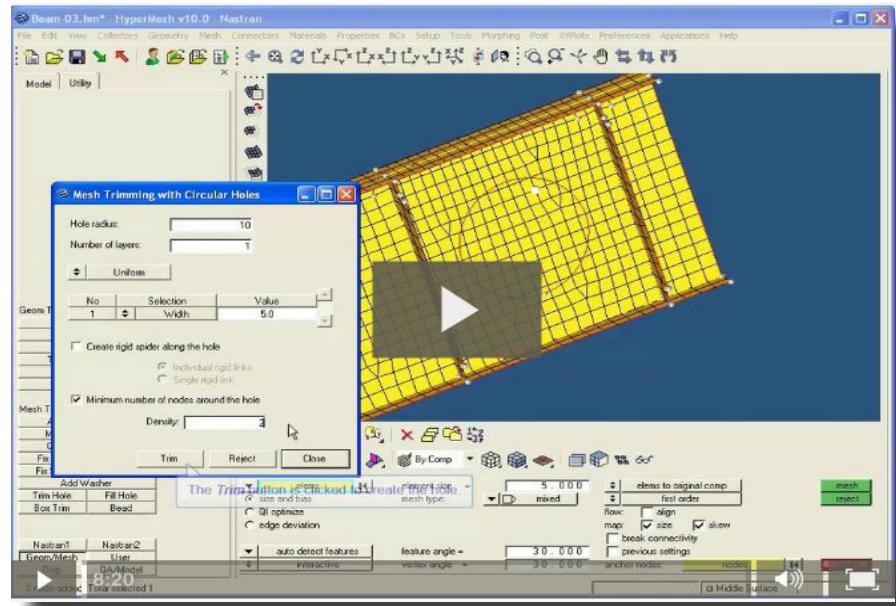
Check element normals (<http://altair-2.wistia.com/medias/dk91yzip0z>)



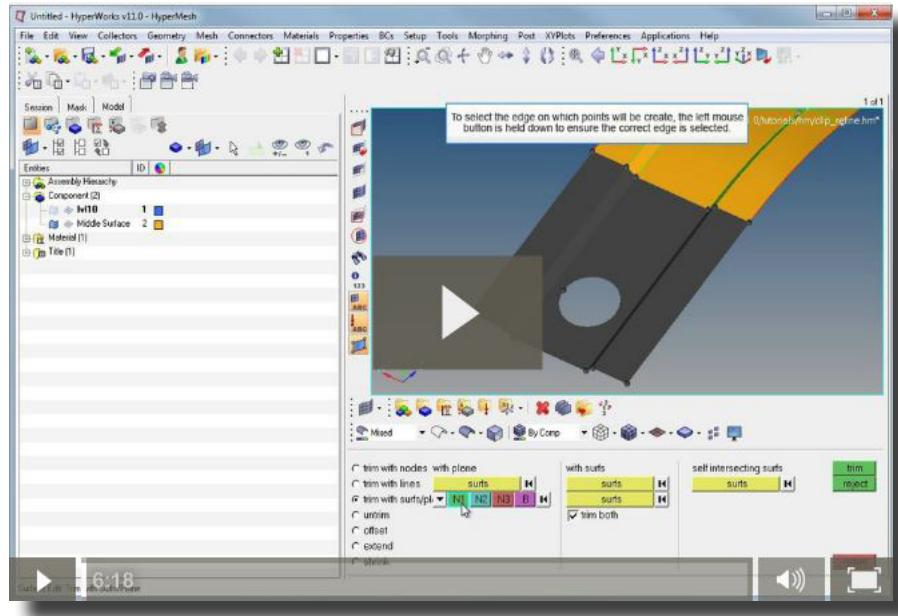
Mesh checking editing (<http://altair-2.wistia.com/medias/ftu42udf77>)



Editing mesh (<http://altair-2.wistia.com/medias/1otl1kftp7>)

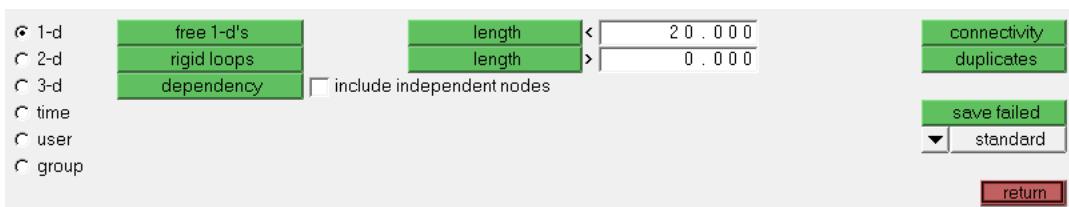


Refining topology (<http://altair-2.wistia.com/medias/2p48j67uav>)



13.11 Student Racing Car Project - Mesh Quality

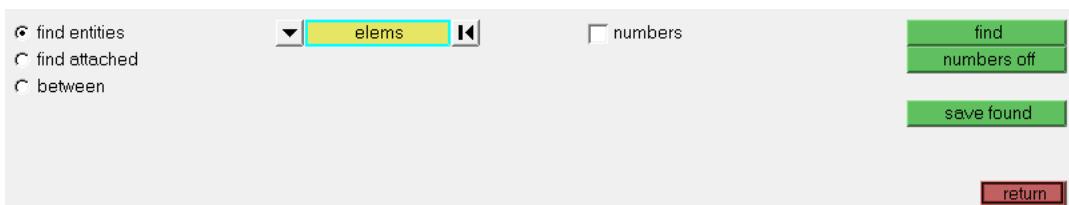
What is the minimum element size? Open the Check Elements panel by clicking on the symbol .



As a “critical” minimum element size (no special reason for this figure) 20 mm is chosen. The check reveals “4 of 372 (1%) failed. The minimum length is 2.72.”

What is causing these small elements? To extract the failed elements, click on “save failed” (right after the check was carried out) which places the failed elements in a dedicated user mark. This results in the following message, “The highlighted elements have been placed in the user mark.”

Then turn off all elements and geometry within the Model Browser (leaving a blank screen). In order to retrieve the failed elements from the user mark activate the Find option  and then click on elems:

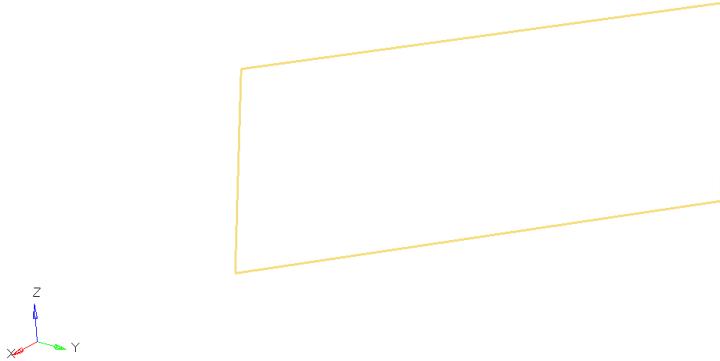


In the extended selection window select retrieve.

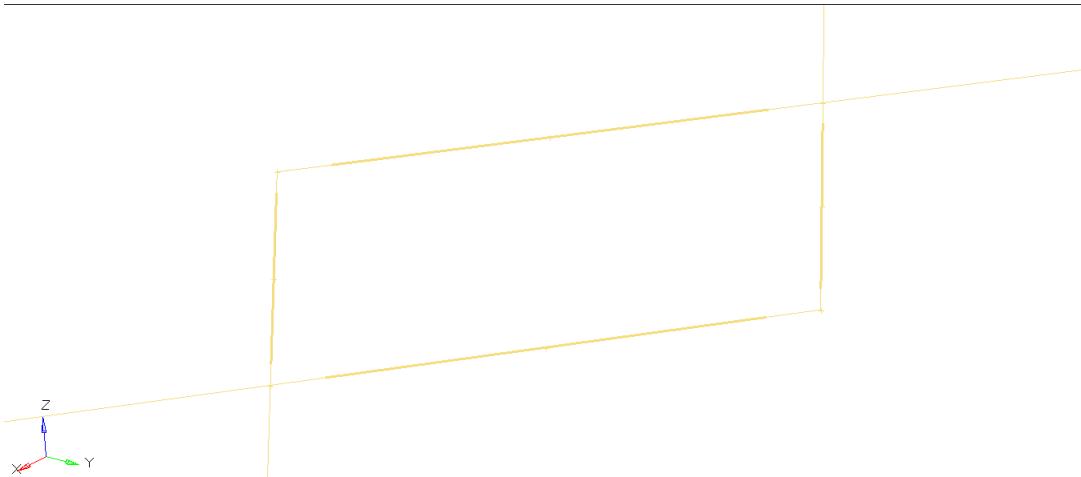
by window	on plane	by width	by geoms	by domains	by laminate
displayed	retrieve	by group	by adjacent	by handles	by path
all	save	duplicate	by attached	by morph vols	by include
reverse	by id	by config	by face	by block	
by collector	by assems	by sets	by outputblock	by ply	



This will upload the failed elements into the screen (you may have to hit to resize the elements)



Adding line information to the display indicates that something with the underlying lines are wrong!



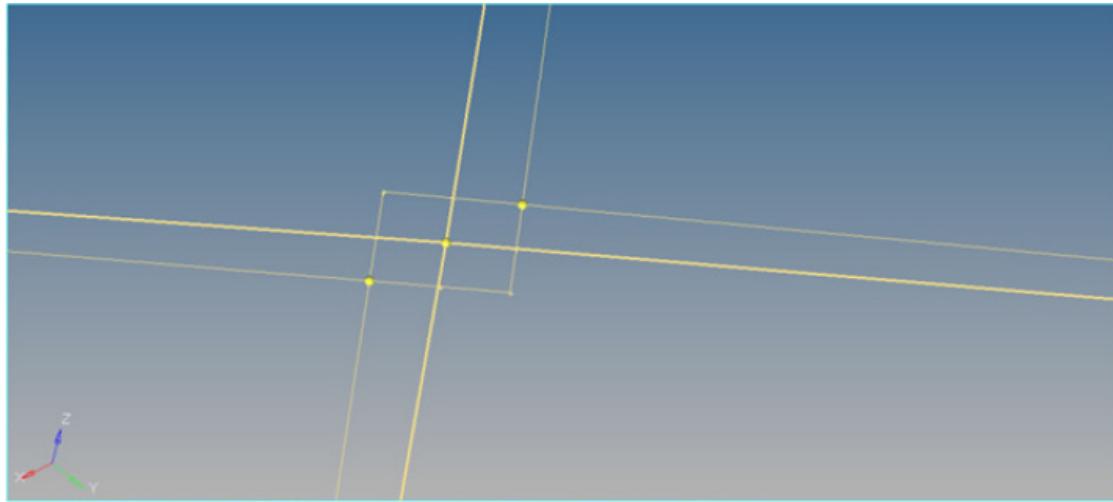
As shown in the image above the CAD lines and hence the CBAR elements are not properly connected. In other words, the squared area is a design flaw and needs to be removed. Depending on the project requirement (for example, meshing with different mesh sizes), you may decide to fix (repair) the problem by rebuilding the lines or by editing the elements directly. For this example, we will fix the elements directly:

First step: deleting the small elements (length < 2.7 mm) (use the Delete panel).



As a result, the 1D elements are not coupled anymore (i.e. there is a physical gap). In order to merge both nodes together, the functionality “replace” is employed (Page 1D > replace):

masses	bars	connectors	line mesh	edit element	<input type="radio"/> Geom
joints	rods	spotweld	linear 1d	split	<input checked="" type="radio"/> 1D
markers	rigids	HyperBeam		replace	<input type="radio"/> 2D
rbe3				detach	<input type="radio"/> 3D
springs				order change	<input type="radio"/> Analysis
gaps			vectors	config edit	<input type="radio"/> Tool
			systems	elem types	<input type="radio"/> Post



The “replace” functionality allows to merge (equivalence) nodes in such a way, that selected nodes move towards each other and meet at their midpoint (half the gap distance). Of course in doing this, we are introducing another inaccuracy into the model.

replace:	<input type="text" value="node"/>	<input type="button" value="I"/>	<input checked="" type="checkbox"/> equivalence	<input type="button" value="reject"/>
with:	<input type="text" value="node"/>	<input type="button" value="I"/>	<input checked="" type="checkbox"/> at mid-point	<input type="button" value="return"/>
			<input type="text" value="total ="/> <input type="text" value="x dist ="/> <input type="text" value="y dist ="/> <input type="text" value="z dist ="/>	

14 Linear Elastic Material Information

This chapter includes material from the book “**Practical Finite Element Analysis**”. It also has been reviewed and has additional material added by Sascha Beuermann.

14.1 Hooke's Law And Two Constants

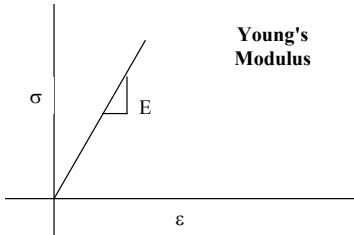
It is common knowledge that for the same force effect (i.e. same stress) you get different strains for different materials. Considering a simple tensile test you have for a lot of materials, for small displacements there is a linear correlation between stress (force per unit area) and strain (elongation per unit length).

$$\sigma = F/A$$

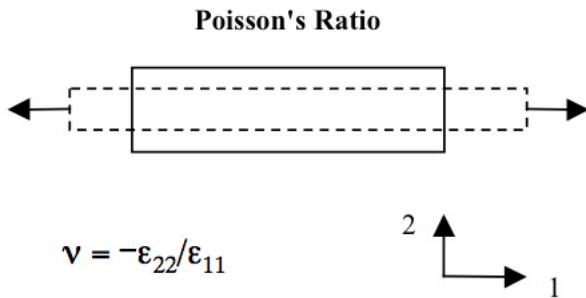
$$\varepsilon = \Delta L/L$$

$$\sigma \sim \varepsilon \Rightarrow \sigma = E\varepsilon$$

with the constant E which is dependent on the material. This equation is called Hooke's Law (Robert Hooke, 1635-1703) and is a material equation for linear elastic behavior. E is called the Modulus of elasticity or Young's Modulus, and the slope of the normal stress-strain curve in the linear elastic domain. It is defined as normal stress / normal strain. Units: e.g. N/mm².



Another phenomenon can be seen at the tensile test. There is not only an elongation in the force direction, but also a contraction in the lateral direction.



For the physical interpretation of ν consider a cube of 1x1x1 mm dimension. Poisson's ratio of 0.30 means that if the cube is elongated by 1 mm, lateral direction contraction would be 0.3 mm. For metals poisson's ratio is between 0.25 to 0.35. The maximum possible value of poisson's ratio is 0.5 (for rubber).

There is also the material parameter G Modulus of rigidity, which is the slope of the shear stress-strain curve in the linear elastic domain. It is defined as shear stress/shear strain. Units: e.g. N/mm²

E, G, and ν are inter-related by the equation:

$$E = 2G(1+\nu)$$

Only two independent material constants are required for a linear static analysis (i.e. E, and ν). Additional data is required for gravity, centrifugal load, and dynamic analysis (material density $\rho = m/V$, mass per unit volume e.g. g/cm³) and for temperature induced stresses or strains (coefficient of thermal expansion $\alpha = \varepsilon/\Delta T = \Delta l/l\Delta T$, expansion/shrinkage per unit length per temperature e.g. 1/K).

Typical values for steel, $\rho = 7.89 \cdot 10^9$ t/mm³ and $\alpha = 1.2 \cdot 10^{-5}$ 1/K, and for aluminum, $\rho = 2.7 \cdot 10^9$ t/mm³ and $\alpha = 2.4 \cdot 10^{-5}$ 1/K.

14.2 Generalized Hooke's Law And 36 Total Constants In The Equation

Hooke's law is familiar to us as $\sigma = E * \varepsilon$.

This equation holds true for isotropic material that is in the linear elastic domain. The general equation of Hooks law for an anisotropic material is:

$$\sigma_{xx} = E_{11} \varepsilon_{xx} + E_{12} \varepsilon_{yy} + E_{13} \varepsilon_{zz} + E_{14} \gamma_{xy} + E_{15} \gamma_{yz} + E_{16} \gamma_{zx}$$

$$\sigma_{yy} = E_{21} \varepsilon_{xx} + E_{22} \varepsilon_{yy} + E_{23} \varepsilon_{zz} + E_{24} \gamma_{xy} + E_{25} \gamma_{yz} + E_{26} \gamma_{zx}$$

$$\sigma_{zz} = E_{31} \varepsilon_{xx} + E_{32} \varepsilon_{yy} + E_{33} \varepsilon_{zz} + E_{34} \gamma_{xy} + E_{35} \gamma_{yz} + E_{36} \gamma_{zx}$$

$$\tau_{xy} = E_{41} \varepsilon_{xx} + E_{42} \varepsilon_{yy} + E_{43} \varepsilon_{zz} + E_{44} \gamma_{xy} + E_{45} \gamma_{yz} + E_{46} \gamma_{zx}$$

$$\tau_{yz} = E_{51} \varepsilon_{xx} + E_{52} \varepsilon_{yy} + E_{53} \varepsilon_{zz} + E_{54} \gamma_{xy} + E_{55} \gamma_{yz} + E_{56} \gamma_{zx}$$

$$\tau_{zx} = E_{61} \varepsilon_{xx} + E_{62} \varepsilon_{yy} + E_{63} \varepsilon_{zz} + E_{64} \gamma_{xy} + E_{65} \gamma_{yz} + E_{66} \gamma_{zx}$$

There are a total of 36 constants ($E_{11}, E_{12}, \dots, E_{66}$) out of which 21 are independent.

14.3 Material Classification

Isotropic	Orthotropic	Anisotropic	Laminates
<p>Iso – same, Tropic - directions</p> <ul style="list-style-type: none"> Properties independent of direction/axes 2 Independent Constants (E, ν) Metals 	<p>Ortho – three, Tropic - directions</p> <ul style="list-style-type: none"> Different properties along 3 axes 9 independent constants Wood, Concrete, rolled metals 	<ul style="list-style-type: none"> Different properties along crystallographic plane 21 independent constants All real life materials are anisotropic only but we simplify them into category of Isotropic and Orthotropic 	<ul style="list-style-type: none"> Two or more materials bonded together in layers. Simplest example is lamination carried out on certificates, Identity cards etc. Mainly used for space applications and these days in automobiles the trend is shifting towards plastics and laminates from metals.

14.4

Material Properties

Material	Elastic Modulus (N/mm ²)	Poisson's Ratio	Density (tonne/mm ³)	Yield strength (N/mm ²)	Ultimate strength (N/mm ²)
Steel	2.10 • 10 ⁵	0.30	7.89 • 10 ⁻⁹	250	420
Cast Iron	1.20 • 10 ⁵	0.28	7.20 • 10 ⁻⁹	85	220
Wrought Iron	1.90 • 10 ⁵	0.30	7.75 • 10 ⁻⁹	210	320
Aluminium	0.70 • 10 ⁵	0.35	2.70 • 10 ⁻⁹	35	90
Aluminium alloy	0.75 • 10 ⁵	0.33	2.79 • 10 ⁻⁹	165	260
Brass	1.10 • 10 ⁵	0.34	8.61 • 10 ⁻⁹	95	280
Bronze	1.20 • 10 ⁵	0.34	8.89 • 10 ⁻⁹	105	210
Copper	1.20 • 10 ⁵	0.34	9.10 • 10 ⁻⁹	70	240
Copper alloy	1.25 • 10 ⁵	0.33	9.75 • 10 ⁻⁹	150	400
Magnesium	0.45 • 10 ⁵	0.35	1.75 • 10 ⁻⁹	70	160
Titanium	1.10 • 10 ⁵	0.33	4.60 • 10 ⁻⁹	120	300
Glass	0.60 • 10 ⁵	0.22	2.50 • 10 ⁻⁹	--	100
Rubber	50	0.49	0.92 • 10 ⁻⁹	4	10
Concrete	0.25 • 10 ⁵	0.15	2.10 • 10 ⁻⁹	--	40

* Above listed properties are approximate and properties as per actual material composition are recommended.

Why mass is in tonne and density tonne /mm³ for N-mm unit system?

$$1 \text{ N} = 1 \text{ kg} \cdot 1 \text{ m} / \text{s}^2 \quad (\text{F} = \text{m} \cdot \text{a}) \quad \Rightarrow \quad 1 \text{ N} = 1 \text{ kg} \cdot 1000 \text{ mm/s}^2$$

$$\Rightarrow 1 \text{ N} = 1000 \text{ kg} \cdot 1 \text{ mm} / \text{s}^2 \quad \Rightarrow \quad 1 \text{ N} = 1 \text{ tonne} \cdot 1 \text{ mm/s}^2$$

Hence when force is specified in N, length in mm, mass must be specified in tonne and density in tonne /mm³

Examples of sets of consistent units (see Chapter 5) for steel are:

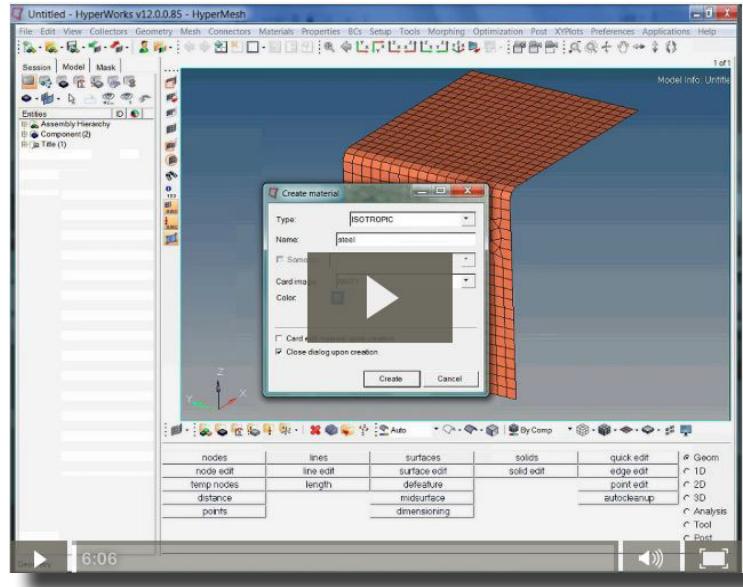
	SI-system	System mm-t-s	System mm-kg-ms
Length unit	meter	millimeter	millimeter
Mass unit	kilogram	tonne	kilogram
Time unit	second	second	millisecond
Force unit	Newton	Newton	kiloNewton
Young's Modulus	210.0e+09	210.0e+03	210.0
Density	7.85e+03	7.95e-09	7.98e-06
Poisson's Ratio	0.3	0.3	0.3

14.5 Linear Elastic Material And Property Tutorials And Videos

Recommended Videos

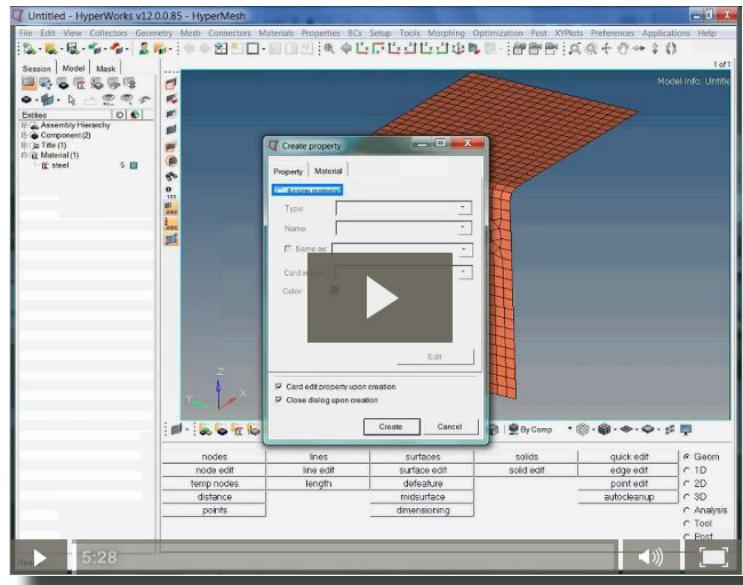
Material definition

(<http://altair-2.wistia.com/medias/f5ns69ht1t>)



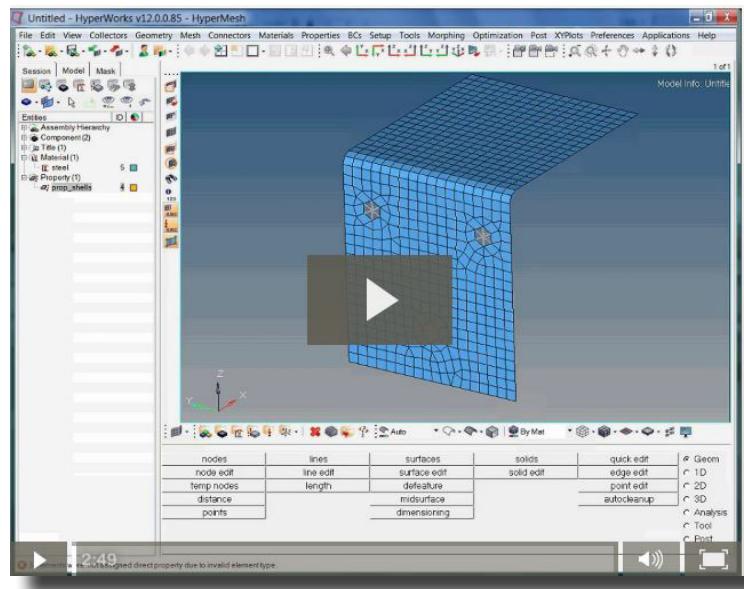
Property definition

(<http://altair-2.wistia.com/medias/wzxc0wm2r2>)



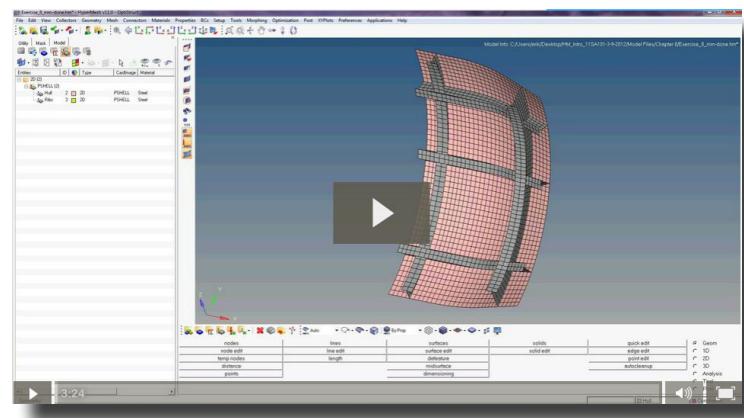
Assign property to elements

(<http://altair-2.wistia.com/medias/93agyi4f0v>)



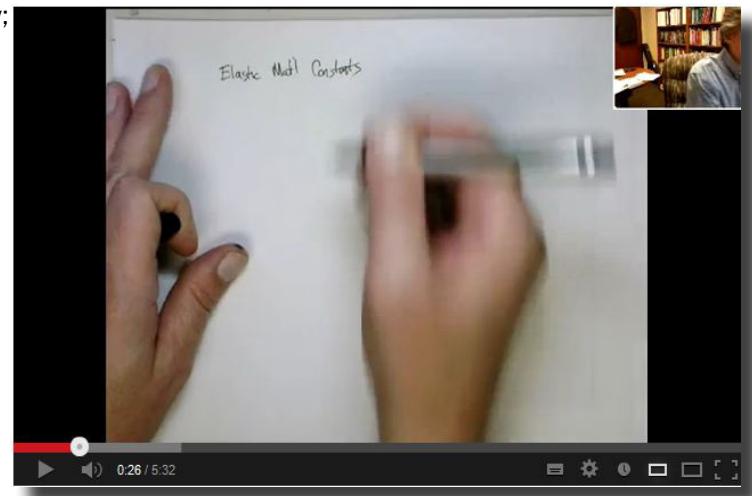
Material property browser

(<http://altair-2.wistia.com/medias/c07ugfgthk>)

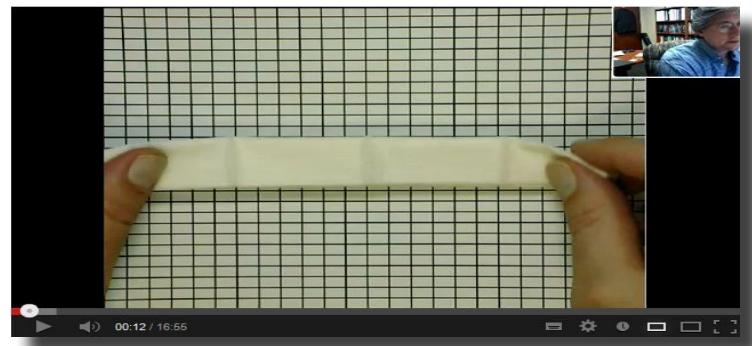


More general aspects (not related to HyperWorks)

Elastic material constant (by Prof. J. Chessa, Texas University; <http://youtu.be/T0XiH8tCv1o>)



Poisson's Ratio (by Prof. J. Chessa, Texas University; <http://youtu.be/VyLSJfUEJSk>)



Lecture on Tensile Test and Hooke's Law; part 1 (by Prof. J. Chessa, Texas University; <http://youtu.be/LKApztxsuU>)



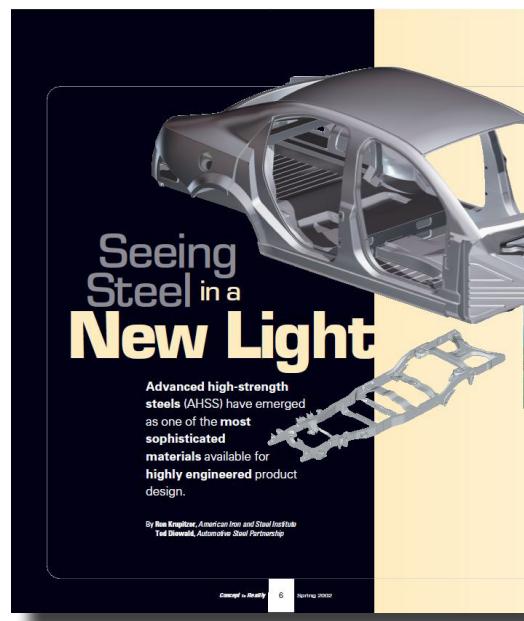
Lecture on Tensile Test and Hooke's Law; part 2 (by Prof. J. Chessa, Texas University; <http://youtu.be/NniY00qlZFQ>)



PDF document

Seeing Steel in a New Light

(http://www.altairuniversity.com/wp-content/uploads/2013/07/Steel_New_Light.pdf)



14.6 Student Racing Car Project - Material Definition

The frame will be built out of steel with the following properties: linear elastic, isotropic and temperature independent, which corresponds to the definition of MAT1 (Card image).

The screenshot shows two parts of the HyperWorks interface. The top part is a table of material properties with columns for Name and Value. The bottom part is a table of material properties with columns for ID, [E], [G], [NU], [RHO], [A], [TREF], and [GE].

Name	Value
Solver Keyword	MAT1
Name	steel
ID	1
Color	
Include File	[Master Model]
Card Image	MAT1
User Comments	Hide In Menu/Export
E	210000.0
G	
NU	0.3
RHO	7.85e-009

	ID	[E]	[G]	[NU]	[RHO]	[A]	[TREF]	[GE]
MAT1	1	2.1e+05		0.300	7.9e-09			
	[ST]	[SC]	[SS]					

Units are: N, mm² and t (tons)

Frame Property Definition & Trouble Shooting

For simplicity reasons the frame will be modeled with CBAR elements referencing a “tube” cross-section with constant properties. The CBAR element properties can be best created with HyperBeam (integrated in HyperMesh). Once the cross section is defined, the cross section information is then stored as a beamsection within a beamsection collector.

The workflow is as:

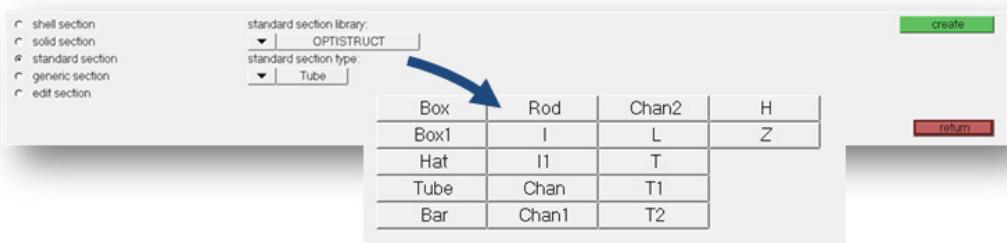
1. Define the required tube cross-section with HyperBeam and store the respective cross section data (area, moment of inertia etc.) in a beamsection collector
2. Generate a property collector with Card Image PBARL. Inside this property collector the previously defined beamsection and a material collector is referenced. Eventually, the property collector is then assigned to all CBAR elements

These steps are explained in some detail next:

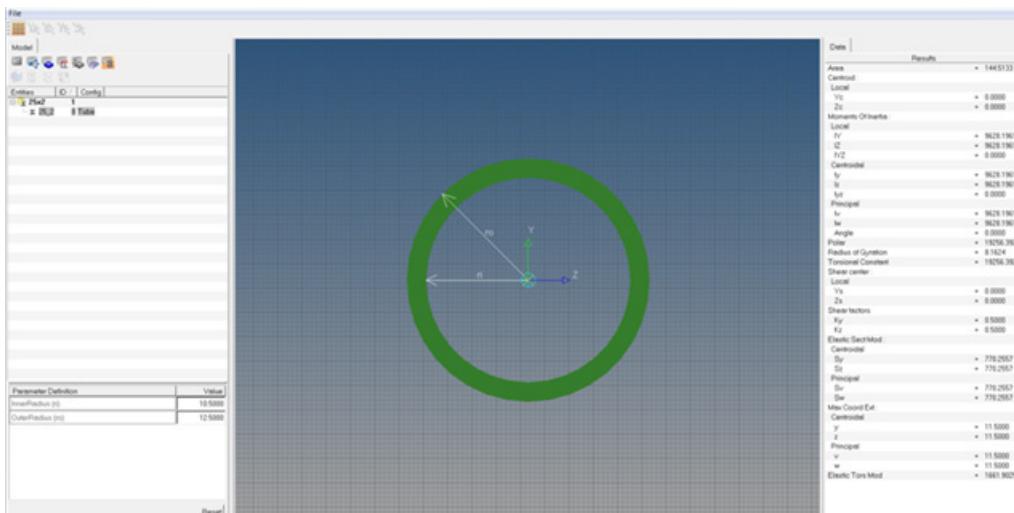
1. Defining The Tube Cross-Section With HyperBeam

HyperBeam can be accessed through the menu bar by selecting Properties > HyperBeam.

Note: As we are using OptiStruct as FEM solver, our lives become easier if we make use of the 1D section library of OptiStruct. Therefore, the standard section library should be OptiStruct, the standard section type should be Tube. Other section do exist of course.



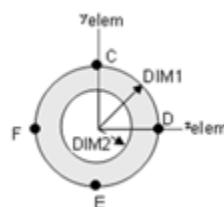
In the graphic editor of HyperBeam, the chosen tube section is displayed.



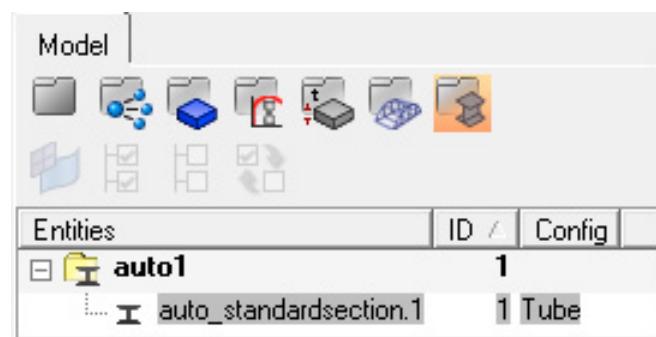
For the base design (it is actually just a “guess”) the tubes radii are assumed to be :

$$r_{\text{out}} = 12.5 \text{ mm}, r_{\text{inside}} = 10.5 \text{ mm}.$$

Of course, the tubes inner (DIM2) and outer (DIM1) radii can be easily changed and adjusted in HyperBeam.



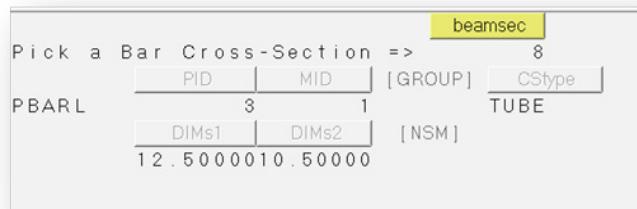
Based on the cross-section, values such as area, moments of inertia etc. are automatically determined and listed in the side bar of HyperBeam. All this information is now readily available as a beamsection stored in a beamsection collector (Note, that different beamsections with different attributes, names and ID's may reside inside a beamsection collector).



Once the definition of the cross section is completed you can return to the HyperMesh GUI by clicking the icon  within the Model Browser.

2. Defining A Property Collector

As before with the material collector, make a right mouse button click in the Model Browser and specify the Card Image PBARL as well as the previously defined material.



The Card image of the property collector (PBRL) includes references to:

- Beamsection (here the ID of the beamsection is 8) of cross section type (CType) TUBE. Inner and outer radii are DIM2 and DIM1
- Material (MID=1)

In the next step, this property collector is assigned to the 1D elements. Hereto, the respective functionality available in the Model Browser is used (right mouse button click on the property and select Assign).

To better visualize the 1D elements, we switch from the Traditional Element Representation  to 3D Element Representation . In addition the element shrink option  is activated.

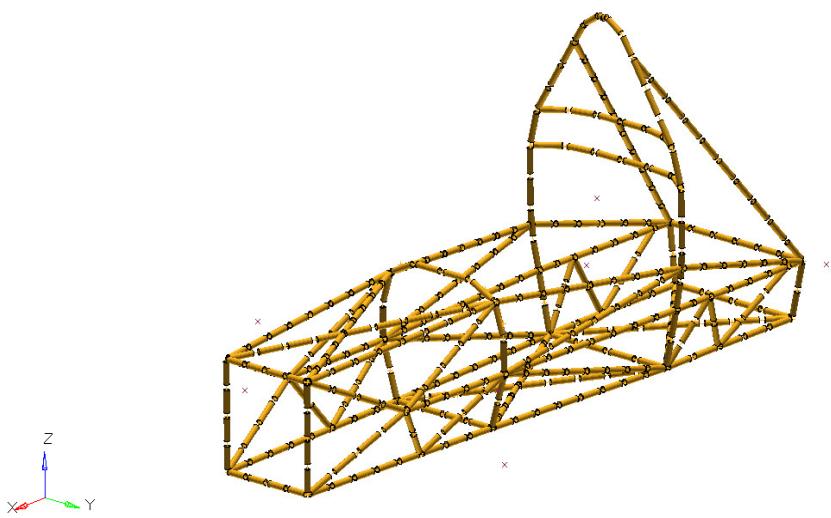


Figure: Car frame meshed with 1D elements (CBARS) with shrink view  on. The x-symbols outside the frame refer to geometry points. These points will be used to model the wheel suspension.

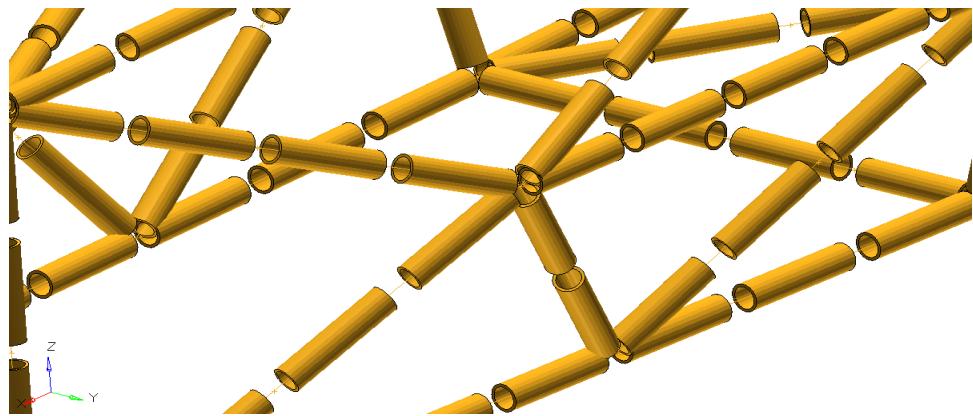
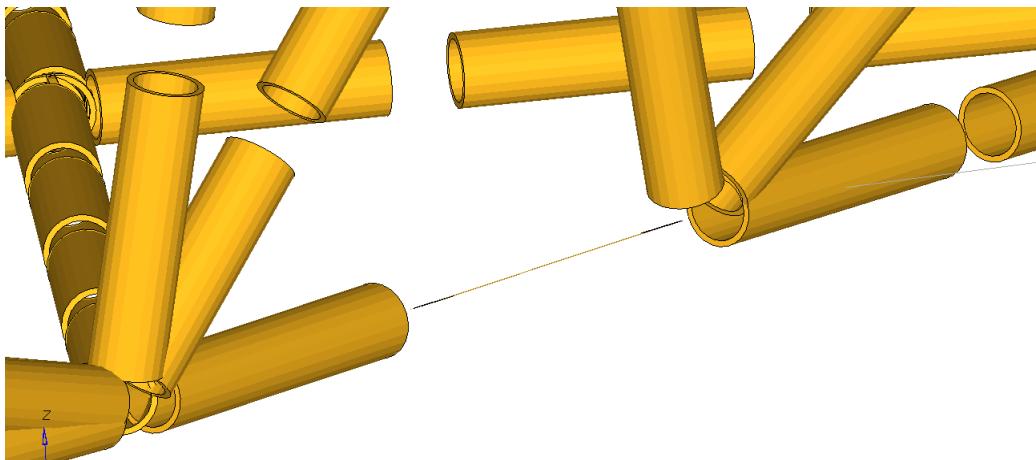


Figure: Detailed view of the meshed frame (shrink view mode is active). A uniform tube cross section is used for simplicity reasons ($r_o=12.5$ mm, $r_i=10.5$ mm).

Trouble Shooting

However, it may happen that some 1D elements are not properly displayed. What is the problem?



In the figure above, the depicted line is actually a CBAR element.

Likely Causes And Ways To Solve This Issue:

1. Make sure that a “property” is assigned to ALL the 1D elements. This can be checked visually in many ways, for instance, by activating the element display mode:



If a property collector is assigned to the 1D elements, they will be displayed with the same color as the property collector.

1. Check the properties of the 1D elements again i.e. review the property collector (in the Model Browser make a right mouse button click on the corresponding property collector and select Card Edit).
2. Check the element orientation.

The element orientation is defined through its local xy-plane. The local element xy-plane in turn is defined through its x-axis (longitudinal direction from node a to b) and a vector v. Basically, one needs to take care of the v vector in order to solve (determine) the element orientation!

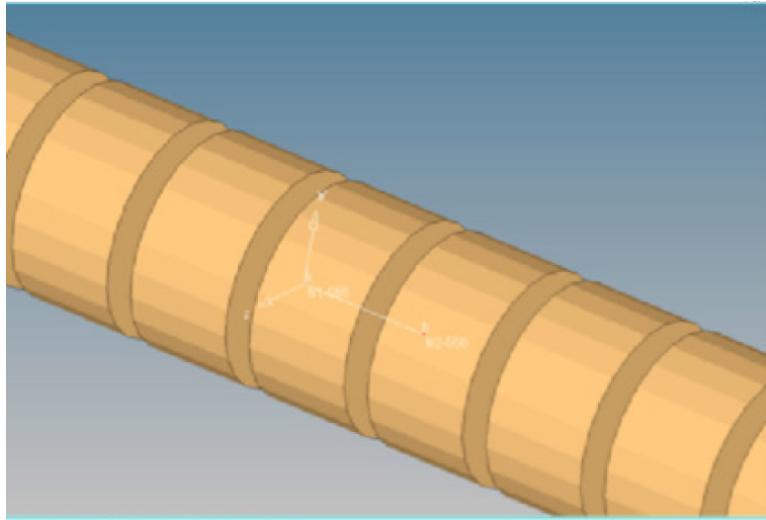
(more information is also available in the help documentation \help\hwsolvers\hwsolvers.htm, then Index > CBEAM or CBAR)

To control (or update) the local element xy-plane and thus the v vector go to the Bars panel by selecting Mesh > Edit > 1D Elements > Bars > Update from the menu bar.

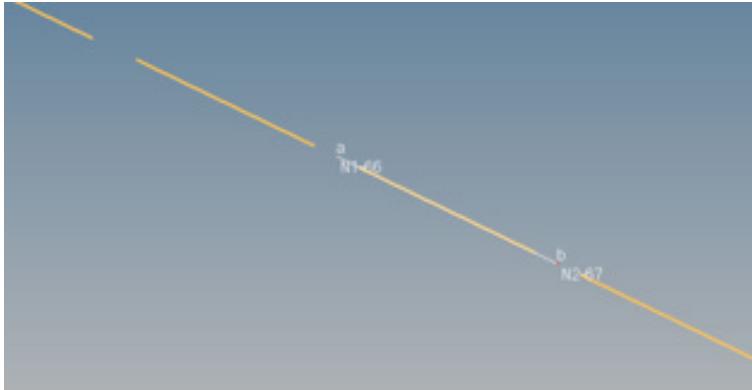
Select any of the displayed 1D elements (in the image the 3D Element Representation  is active).

As shown above this will display not only the local element coordinate system but also the components of the vector v (with respect to the global coordinate system). Here, the vector v parallels the global y-direction(y comp = 1). The element xy-plane thus parallels the global xy-plane.

In the next step, we are reviewing the element orientation of a 1D element which is not properly displayed (cross-sections are not shown).



Just repeat the “review steps” from before:



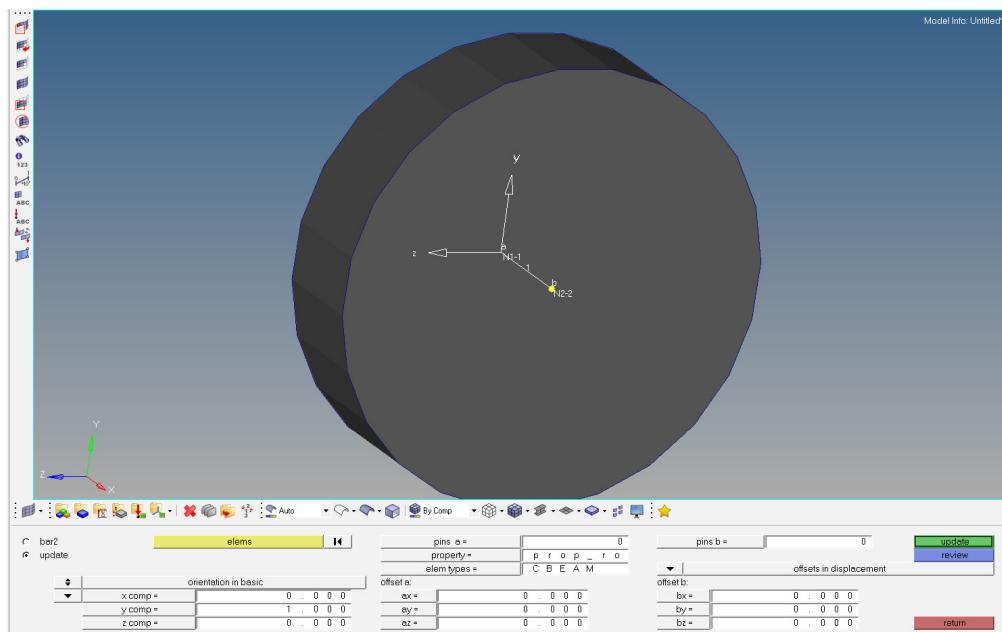
This time the location of node a and b is shown only. In the panel shown below it becomes apparent that the components of the v vector are all zero.

<input type="radio"/> bar2	<input checked="" type="radio"/> update	elements <input type="button" value="H"/>	pins a = <input type="text" value="0"/>	pins b = <input type="text" value="0"/>	<input type="button" value="update"/>
			property = <input type="text" value="p r o p _ r o"/>	offsets in displacement	<input type="button" value="review"/>
			elem types = <input type="text" value="C B E A M"/>	offset a:	
		orientation in basic	ox = <input type="text" value="0 . 0 0 0"/>	bx = <input type="text" value="0 . 0 0 0"/>	
			oy = <input type="text" value="0 . 0 0 0"/>	by = <input type="text" value="0 . 0 0 0"/>	
			oz = <input type="text" value="0 . 0 0 0"/>	bz = <input type="text" value="0 . 0 0 0"/>	
<input type="button" value="return"/>					

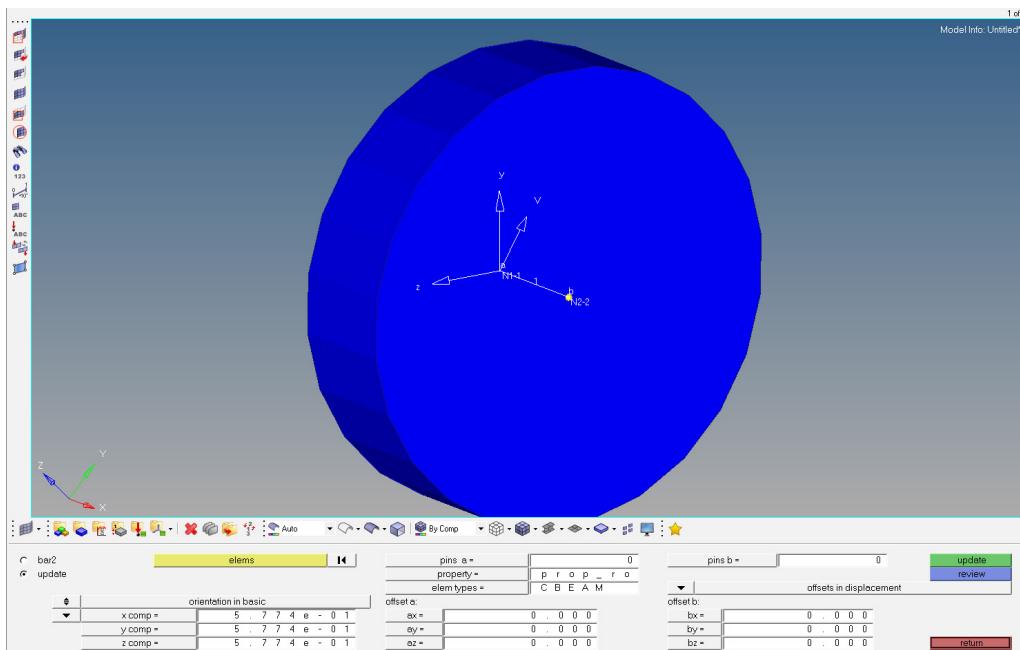
The good thing: the components of vector v can be corrected/updated at any time.

Select a single or all elements, then specify the component of the vector v with respect to the global (basic) coordinate system. In

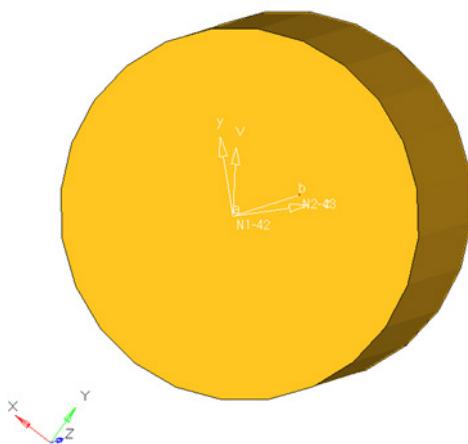
case the y-component of vector v is set to y=1 (global y), both, vector v and local y-direction of the element coincide. In the image below the local y direction and v vector are superimposed on each other.



In the image below, the vector v doesn't coincide with any of the global components. Now it is quite apparent, that vector v and local x-axis together specify the local xy-plane of the element.



The same element is viewed from a different direction



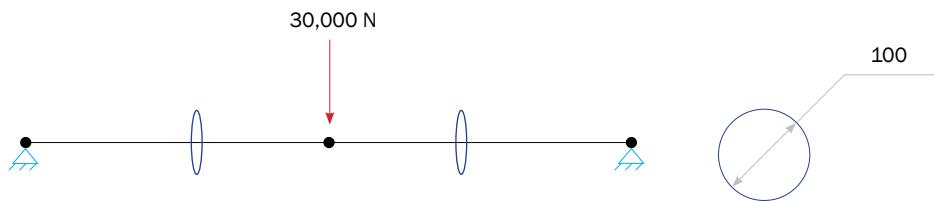
15 Boundary Conditions And Loads

This chapter includes material from the book “**Practical Finite Element Analysis**”. It also has been reviewed and has additional material added by Matthias Goelke.

15.1 Boundary Conditions

Example: Simply supported beam

Note: Even though you may not be able to start an analysis on your own yet, this example provides an interesting insight into the challenging topic of boundary conditions.



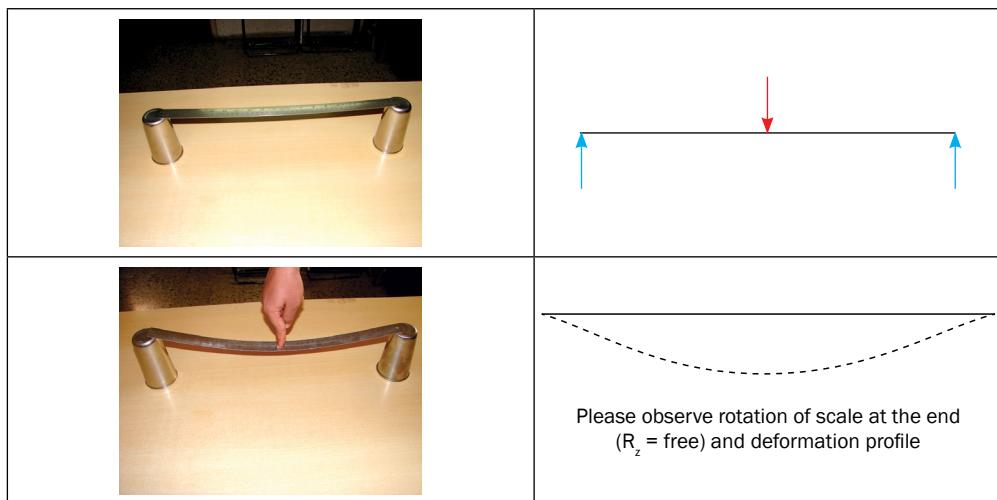
Create 3 nodes and 2 beam elements. Apply a force on the central node and the appropriate boundary conditions on the end nodes. Solve the problem and compare the results. If the applied boundary conditions are correct, then the results would match with following values:

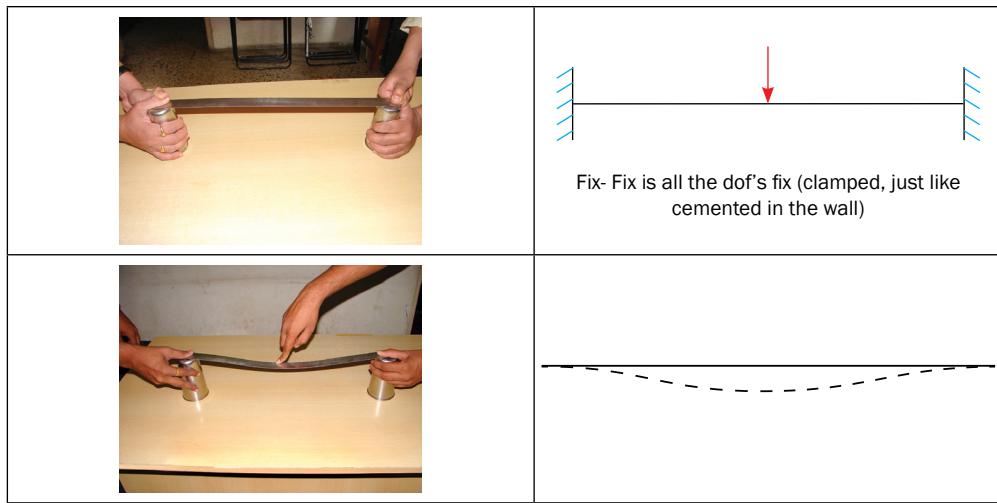
$$\delta = (1/48) * \{PL^3/EI\} \quad \text{and} \quad \sigma = M*y/I \quad (M = PL/4)$$

If your results do not match with the above values, then try following formulas:

$$\delta = (2/384) * \{PL^3/EI\} \quad \text{and} \quad \sigma = M*y/I \quad (M = PL/8)$$

If your answer matches with the 2nd set of formulas, then it means the fix-fix boundary conditions have been applied instead of the simply supported.





	Stress N/mm ²	Displacement mm
Simply supported	76.43	0.64
Fix-Fix	38.22	0.16

The difference between the above two boundary conditions is simply because of one dof, the rotation about the z axis (free in case of simply supported and fixed for fix-fix). There is a drastic change in the stress and displacement results. The stress differs by a factor of 2 and displacements by a factor of 4. Now imagine a real life steel structure which is simply supported, being modelled by a CAE engineer as fix-fix and the analysis shows that stress = 200 N/mm² and displacement = 2.5 mm. Since the magnitude of the stress is less than the yield stress (250 N/mm²), a colorful impressive report with a wonderful power point presentation supported by animation files, would be submitted to a design engineer with the conclusion that the design is safe. But in real life, since it is simply supported, the magnitude of stress will be double (400 N/mm²) and the displacement 4 times (10 mm) and the structure will fail soon.....

Boundary conditions take least amount of time to define, but it is the most important step in FEA. Typically, the brick meshing of a 4 cylinder Engine block requires 300 hours and defining the boundary condition requires 8 hours. 300 hours hard work would go in vain if the boundary conditions are not applied properly.

How To Apply The Correct Boundary Conditions? Who Can Teach Us Quickly?

The application of boundary conditions is a tricky subject and is based on pure engineering judgment, skill, and common sense.

No university, consultant, or book can teach these things. This is something to be learned from practical and field experience. 100% involvement, thinking about the behavior of the component due to various forces, and checking after every iteration whether the FE model is behaving in the same way as in reality (displacement plots and strain gauge results for static analysis or natural

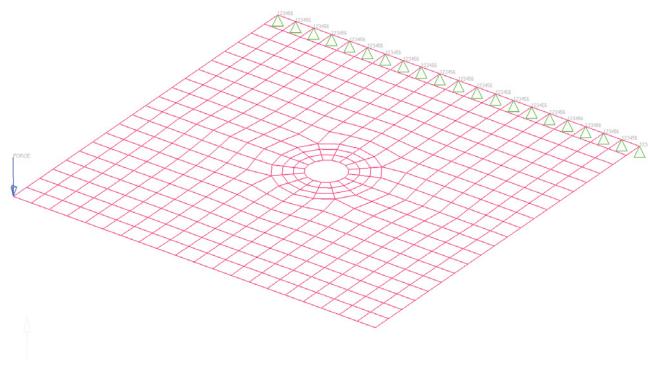
frequencies and response comparison for dynamic analysis, etc.). If the model is not behaving correctly, then modifications in the boundary conditions are needed. Discussions with experienced CAE engineers, field engineers, and testing department engineers are also needed. In short, making mistakes and reworking and modifying the FE model again and again until satisfactory correlation is achieved is the true teacher.

In HyperMesh, boundary conditions are stored within what are called load collectors. Load collectors may be created using the right click context menu in the Model Browser (Create > Load Collector).

Quite often (especially at the beginning) a load collector is needed for the constraints (also called SPC – Single Point Constraints) and a second one is needed for the forces and/or pressures. Keep in mind, you can place any constraints (e.g. nodes constraint) with respect to dof 1, or nodes with constraints dof123, etc. in a single load collector. The same rule applies for forces/pressures. They are stored within a single load collector regardless of their orientation and magnitude.

In the following you will learn some basic principles about the way forces may be applied to a structure.

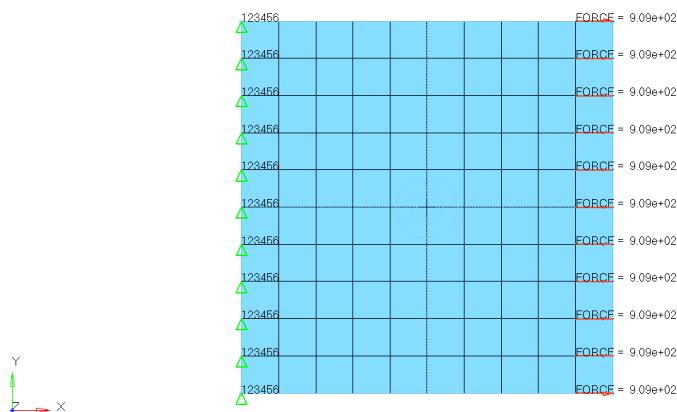
1. Concentrated load (at a point or single node)



Applying forces to single nodes may cause irritating effects, especially while looking at the stresses in this area. Typically concentrated loads (i.e. forces on a single node) impose high stress gradients. Even though the high stresses are correct (i.e. force applied to an infinitesimal small area) one needs to ask whether this kind of loading is reasonable at all? In other words, which real-life loading scenario is represented in the model?

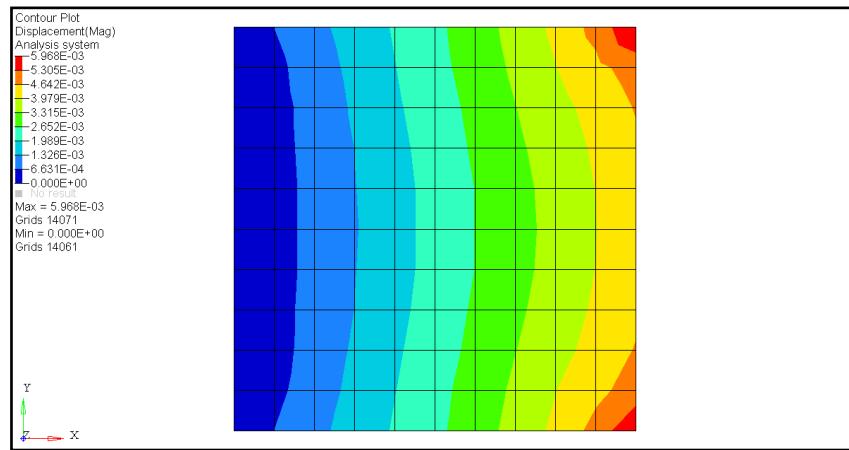
Therefore, forces are commonly applied as distributed loads, namely line loads, and surface loads which are “closer” to reality.

2. Force On Line Or Edge



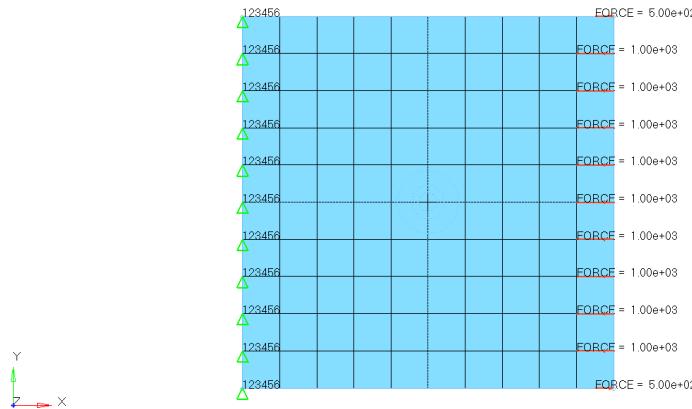
In the above figure, a plate subjected to 10.000 N. The force is equally applied to all (11) nodes at the model edge. Note that the forces at the corner will act only on ½ of the element edge.

The figure below is a displacement contour plot. Note the red “hotspots” located in the corners of the plate. The local displacement maximus are imposed by boundary effects (i.e. the forces applied to the corner nodes act only on $\frac{1}{2}$ of the element edge), however

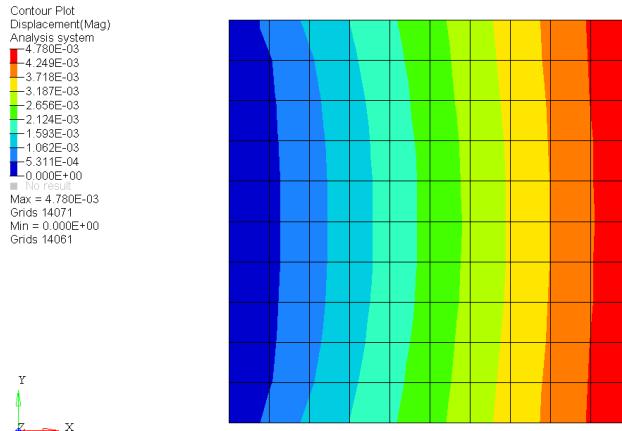


we applied a constant magnitude along the plate's edge.

In the example below, the plate is also subjected to 10.000 N. This time the forces at the corners are just $\frac{1}{2}$ the magnitude of the other applied forces.

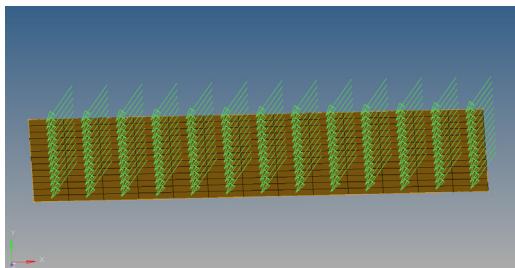


Quite apparently, the displacements are now more evenly distributed (see figure below).

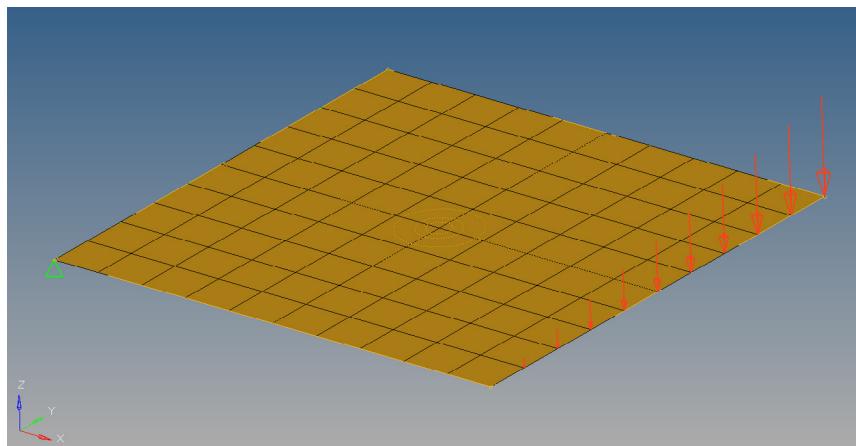


3. Traction (Or “Oblique” Pressure)

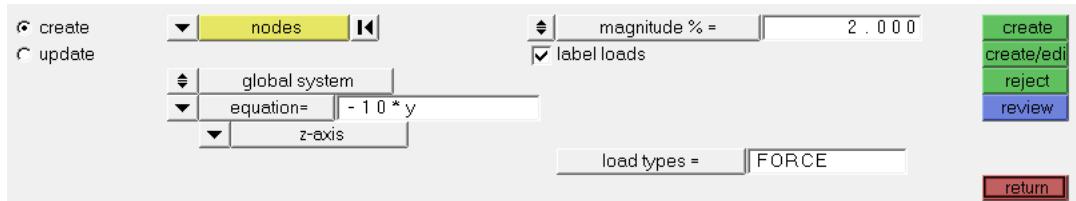
Traction is a force acting on an area in any direction other than the normal direction. A force acting normal to an area is known as pressure.



4. Distributed Load (Force Varying As Equation)

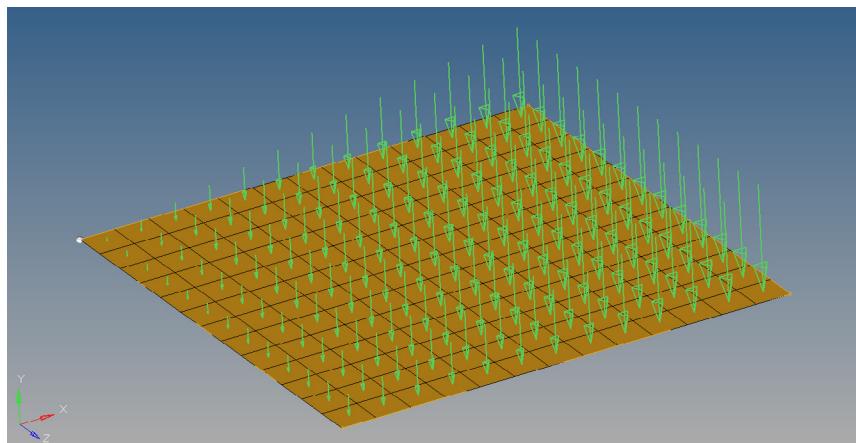


How to apply a force with “variable” magnitude?



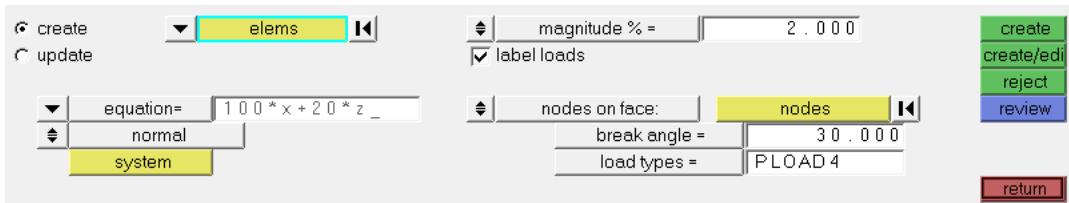
Distributed loads (varying with respects to the coordinates of the nodes or elements) can be applied by means of an equation. In the displayed example, the magnitude of the applied forces varies with respect to the nodal y-coordinate (i.e. the force is acting in negative z-direction and increases along positive y-direction by a factor of 10 respectively).

5. Pressure And Vacuum



In the image above, a distributed load (pressure) is shown. The origin of the plate is located at the highlighted node in the left upper corner.

How to apply a pressure with “variable” magnitude?



In the example above, the magnitude of the applied pressure depends on the x- and z-direction of the elements centroid.

6. Hydrostatic Pressure

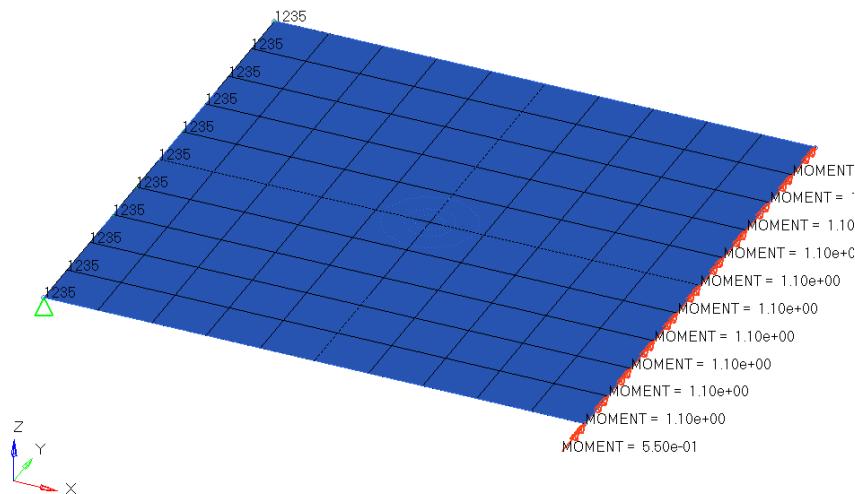
Civil engineering applications: Dam design. Mechanical engineering applications: Vessels / tanks containing liquid.

Hydrostatic pressure is zero at the top surface of the liquid and is maximum ($= \rho * g * h$) at the bottom surface. It varies linearly as shown in the following figure:

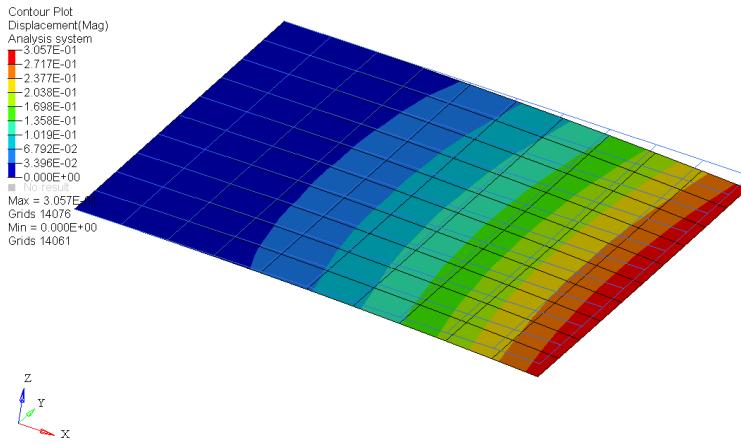
The hydrostatic pressure is applied taking into account the element centroids location (vertical position, h).

7. Bending Moments

The convention for representing a force is a single arrow (\rightarrow), pointed towards the direction of the force . A moment is represented by a double arrow, where the direction of the moment is decided by the right hand rule.

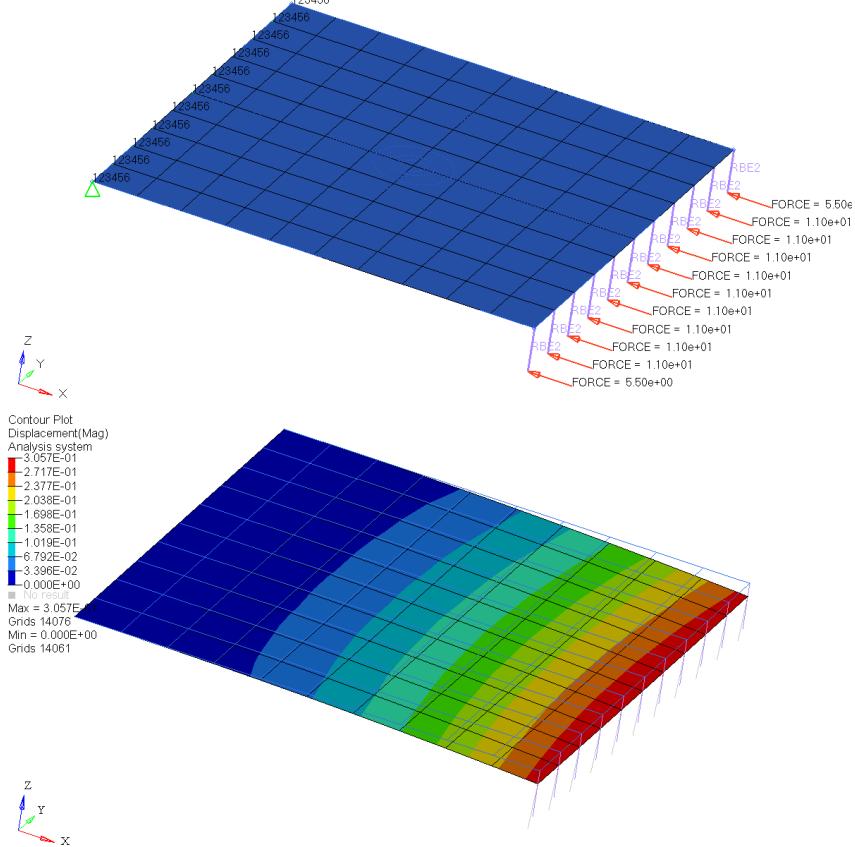


The nodes along the plate's edge are subjected to moments. As a consequence, the nodes will tend to rotate with respect to the y-axis (dof 5).



In the image above, the plate is subjected to moments along its edge on the right. The displacements are scaled by 100 and the initial model is displayed as wire frame.

The moment applied to the nodes in the figure above can also be modeled by adding rigid elements (RBE2) to each node which are then subjected to corresponding forces. In this example, the RBE2 would be oriented in the z-direction and subjected to a force acting in x-direction as shown in the figures below.

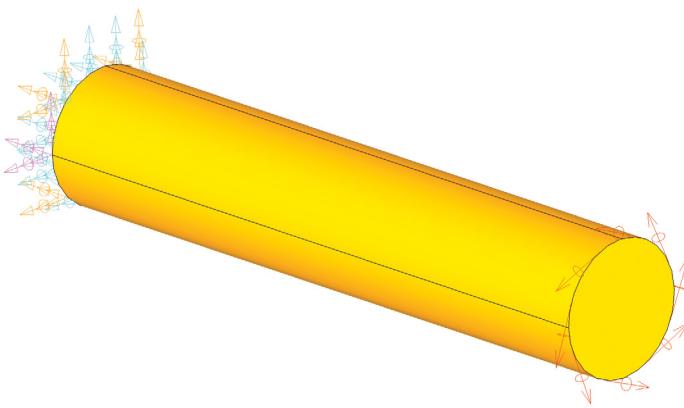


While postprocessing the results make sure that the RBE2 results (i.e. nodal position) are not postprocessed (just display the displacements of the shell elements).

8. Torque

What is torque? Are torque and bending moments different?

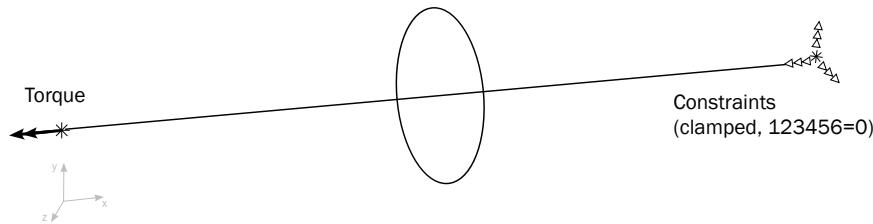
Torque is a bending moment applied parallel to the axis of a shaft (M_x).



Torque or M_x causes shear stresses and angular deformation, while the effect of the other two moments (M_y , M_z) is the normal stress and longitudinal deformation.

How to decide the direction of torque (clockwise or anticlockwise)

It is based on the right hand rule. Point the thumb of your right hand towards the arrow direction. The direction of your fingers indicate the direction of torque.



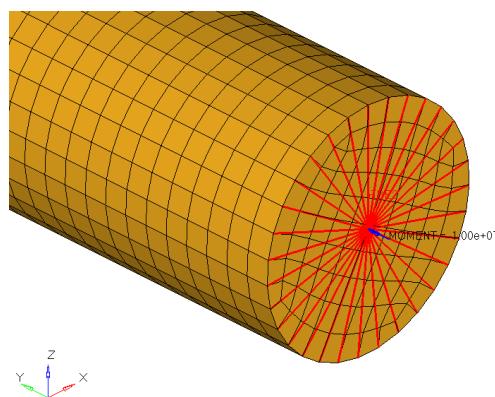
How to apply torque for solid elements (brick /tetra)

As solid elements have no rotational stiffness at the grid points (only 3 translational dofs), a common mistake is to apply torques and moments to the grid points of solid elements directly.

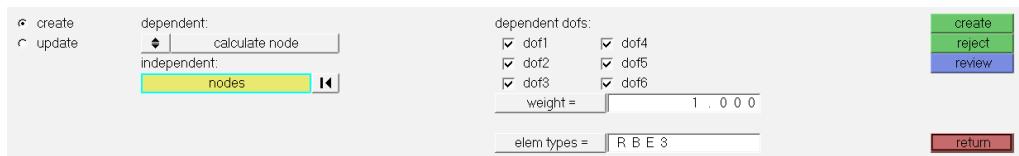
The correct way to apply a moment to a solid model is to use an RBE2 or RBE3 rigid-body element. The rigid-body element distributes an applied moment into the solid element model as forces.

Rigid Element Connection RBE2

A center node is connected to the outer edge nodes using a rigid element (RBE2). The torque is then applied at the center node.

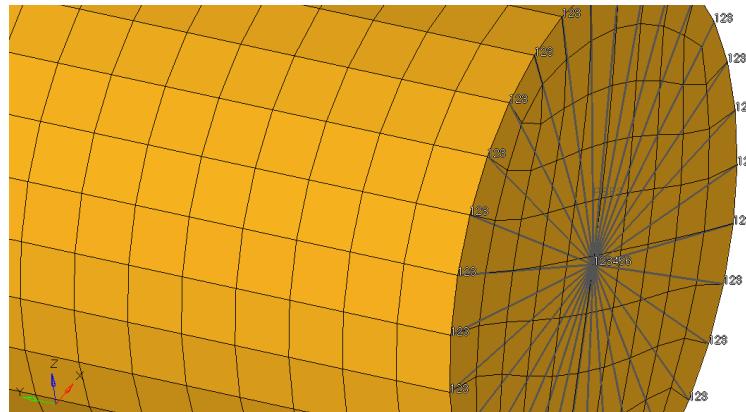


Alternatively, you may use an RBE3 elements instead



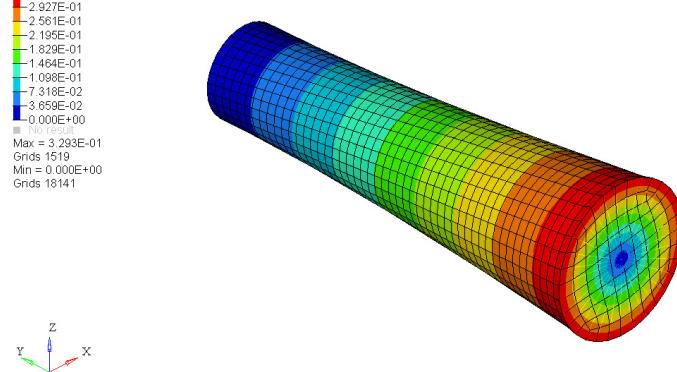
Select the nodes at the outer contour of the shaft as independent nodes. The dependent node may then be determined automatically. It is pretty easy.

However, care needs to be taken with respect to the dof's being referenced. The node of the solid shaft possess translational degrees of freedom only (dof 123). The dependent node also allows for rotational displacements (dof 123456). If the rotational dof (in this example dof 5; rotation y-axis) of the dependent node is not "activated", the moment will not be transferred to the independent nodes.



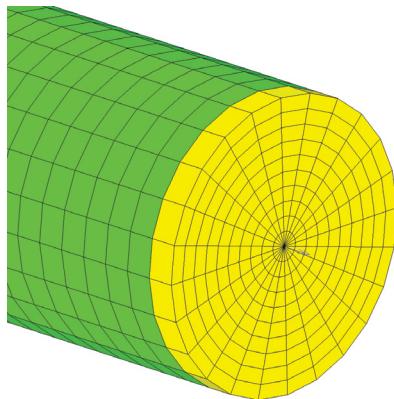
Contour Plot
Displacement(Mag)
Analysis system
-3.293E-01
-2.927E-01
-2.561E-01
-2.195E-01
-1.829E-01
-1.464E-01
-1.098E-01
-7.316E-02
-3.659E-02
0.000E+00

Max = 3.293E-01
Grids 1519
Min = 0.000E+00
Grids 18141



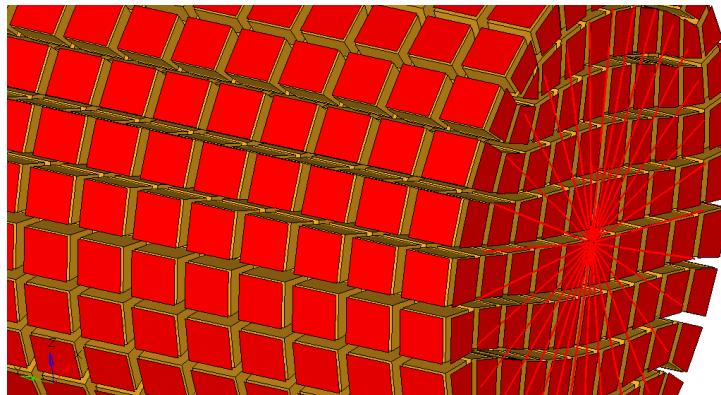
Shell Element Coating

On the brick/tetra element outer face additional quad/tria (2D) elements coating the solid elements are created. The thickness of these shell element should be negligible (so that it would not affect the results). Moment could now be applied on all the face nodes (moment per node = total moment / no. of nodes on the face).



The shell element coating can be easily created within HyperMesh. Create faces using the Faces panel. This panel can be accessed through the toolbar icon which is displayed using View > Toolbars > Checks.

The faces (nothing more than 2D plot elements) are automatically created and stored in a component collector named ^faces. All you need to do is treat these elements as regular elements (i.e. rename the component collector, assign materials and properties).



In the figure above, the elements of the shaft are displayed by means of the shrink element command, . The orange elements are the hexahedral elements (3D), and the red elements are the 2D elements placed on the free faces of the 3D elements.

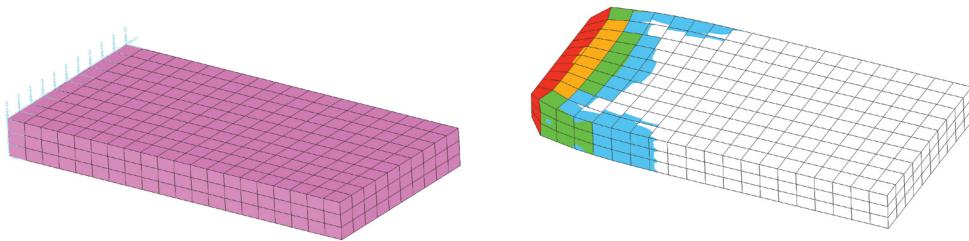
9. Temperature Loading

Suppose a metallic ruler is lying on the ground freely as shown in the figure below. If the temperature of the room is increased to 50 degrees, would there be any stress in the ruler due to temperature?



There will be no stress in the ruler. It will just expand (thermal strain) due to the higher temperature. Stress is caused only when

there is a hindrance or resistance to deformation. Consider another case, this time one end of the metallic object is fixed on a rigid wall (non conductive material). Now if the temperature is increased, it will produce thermal stress (at the fixed end) as shown below.



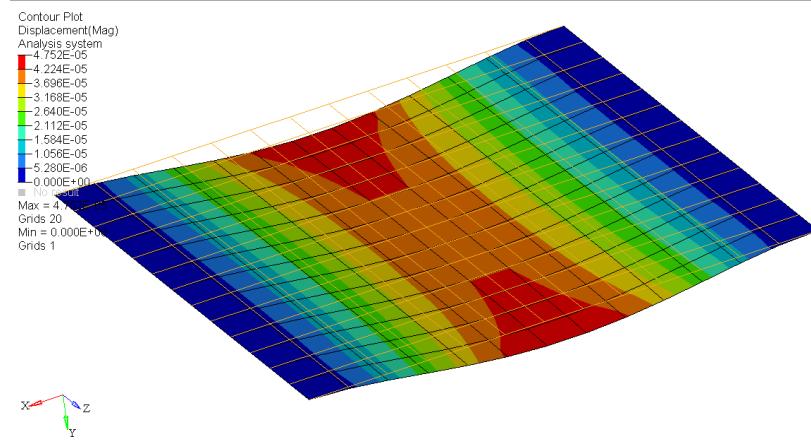
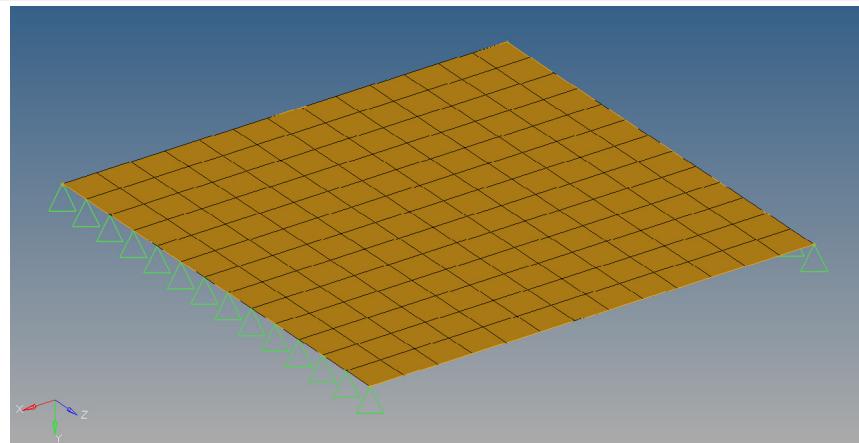
For thermal stress calculations, the input data needed is the temperature value on nodes, the ambient temperature, thermal conductivity, and the coefficient of linear thermal expansion.

10. Gravity Loading

Specify direction of gravity and material density.

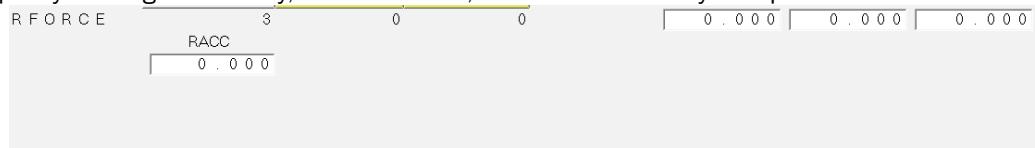
A load collector with the card image GRAV is needed (here the FEM Solver OptiStruct is chosen as the User Profile in HyperMesh). Please keep your unit system in mind.

	SID	CID	G	N1	N2	N3
GRAV	1	0	9 8 1 0 . 0 0	0 . 0 0 0	1 . 0 0 0	0 . 0 0 0



11. Centrifugal Load

The user has to specify the angular velocity, axis of rotation, and material density as input data.



The RFORCE Card Image defines a static loading condition due to a centrifugal force field.

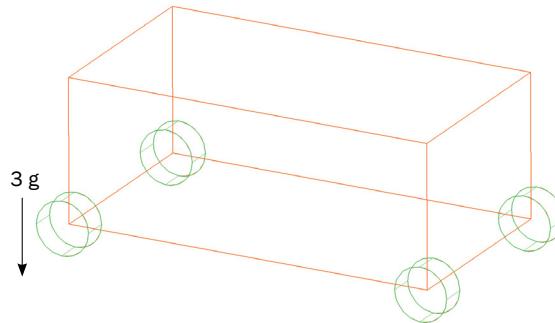
12. 'G' Values (General Rules) For Full Vehicle Analysis

- Vertical acceleration (Impact due to wheel passing over speed braker or pot holes): 3g
- Lateral acceleration (Cornering force, acts when vehicle takes a turn on curves): 0.5 to 1 g
- Axial acceleration (Braking or sudden acceleration): 0.5 to 1 g

13. One Wheel In Ditch

The FE model should include all of the components (non critical components could be represented by a concentrated mass). The mass of the vehicle and FE model mass, as well as actual wheel vehicle reactions and FE model wheel reactions, should match.

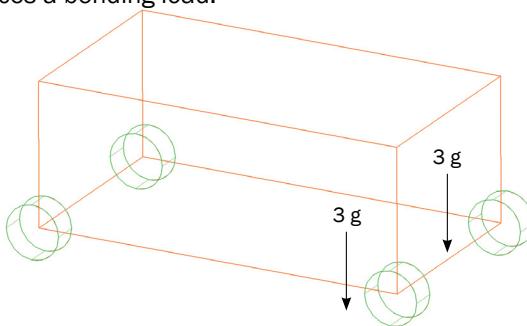
While applying a constraint, the vertical dof of the wheel, which is in considered in the ditch, should be released. The appropriate constraints should be applied on the other wheels so as to avoid rigid body motion. Specify the gravity direction as downward and magnitude = 3×9810 mm/sec²



Another simple but approximate approach (since most of the time either we do not have all the CAD data for the entire vehicle or sufficient time for a detailed FE model) is to apply 3 times the reaction force on the wheel which is in the ditch. Suppose the wheel reaction (as per test data) is 1000 N. Therefore, apply 3×1000 or 3000 N force in the vertically upward direction and constraint the other three wheels enough to avoid rigid body motion. This approach works well for relative design (for comparison of two designs).

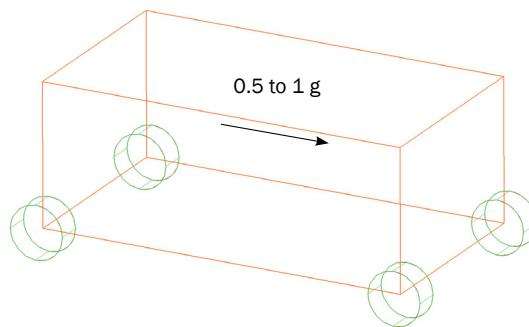
14. Two Wheels In Ditch

Same as discussed above, except now instead of one wheel, two wheels are in the ditch. One wheel in the ditch causes twisting, while two wheels in the ditch produces a bending load.



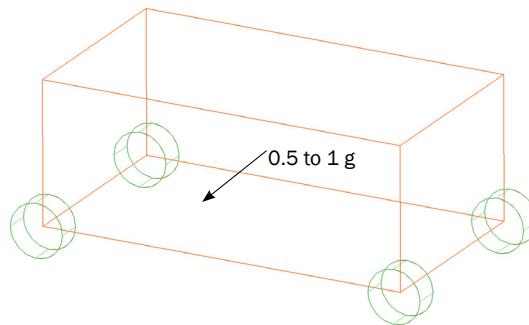
15. Braking

Linear acceleration (or gravity) along the axial direction (opposite to vehicle motion) = 0.5 to 1 g.



16. Cornering

Linear acceleration along the lateral direction = 0.5 to 1 g.



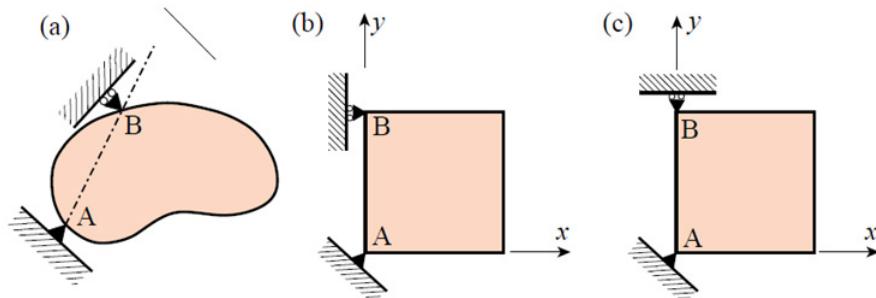
15.2 How To Apply Constraints

A beginner may find it difficult to apply boundary conditions, and in particular, constraints. Everyone who starts a career in CAE faces the following two basic questions:

- i) For a single component analysis, should forces and constraints be applied on the individual component (as per the free body diagram) or should the surrounding components also be considered.
- ii) At what location and how many dof should be constrained.

Constraints (supports) are used to restrain structures against relative rigid body motions.

Supporting Two Dimensional Bodies



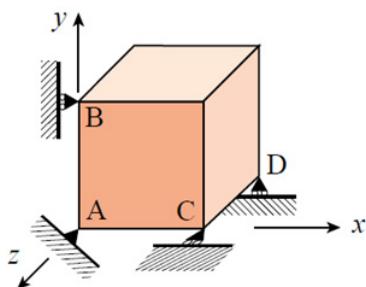
The figure above depicts two-dimensional bodies that move in the plane of the paper (taken from: <http://www.colorado.edu/engineering/CAS/courses.d/IFEM.d/IFEM.Ch07.d/IFEM.Ch07.pdf>).

If a body is not restrained, an applied load will cause infinite displacements (i.e. the FEM program will report a rigid body motion and will abort the run with an error message). Hence, regardless of loading conditions, the body must be restrained against two translations along x and y , and one rotation about z . Thus the minimum number of constraints that has to be imposed in two dimensions is three.

In figure (a) above, the constraint at A fixes (pins) the body with respect to translational displacements, whereas the constrain at B, together with A, provides rotational restraint. This body is free to distort in any manner without the supports imposing any deformation constraints.

Figure (b) is a simplification of figure (a). Here the line AB is parallel to the global y -axis. The x and y translations at point A, and the x translation at point B are restrained, respectively. If the roller support at B is modified as in figure (c), a rotational motion about point A is possible (i.e. the rolling direction is normal to AB). This will result in a singular modified stiffness matrix (i.e. rigid body motion).

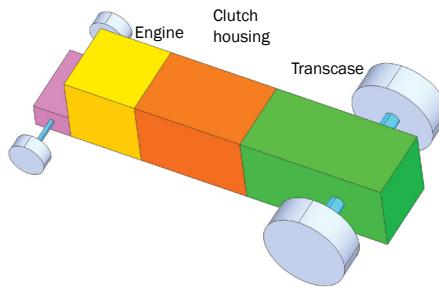
Supporting Three Dimensional Bodies



The figure above (taken from: <http://www.colorado.edu/engineering/CAS/courses.d/IFEM.d/IFEM.Ch07.d/IFEM.Ch07.pdf>) illustrates the extension of the freedom restraining concept to three dimensions. The minimal number of freedoms that have to be constrained is now six and many combinations are possible.

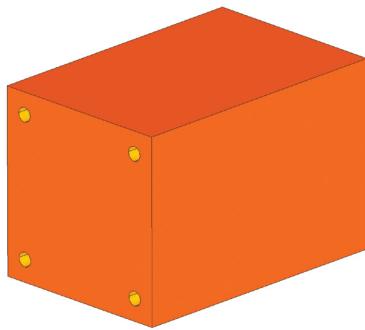
In the example above, all three degrees of freedom at point A have been fixed. This prevents all rigid body translations, and leaves three rotations to be taken care of. The x displacement component at point B is constrained to prevent rotation about z , and the z component is fixed at point C to prevent rotation about y . The y component is constrained at point D to prevent rotation about x .

1. Clutch Housing Analysis



The aim is to analyze (only) the clutch housing. The clutch housing is connected to the engine and transmission case by bolts. There are 2 possibilities for analysis:

Approach 1) Only the clutch housing is considered for the analysis. Therefore, apply forces and moments as per the free body diagram and constrain the bolt holes on both the side faces (all dof = fix).

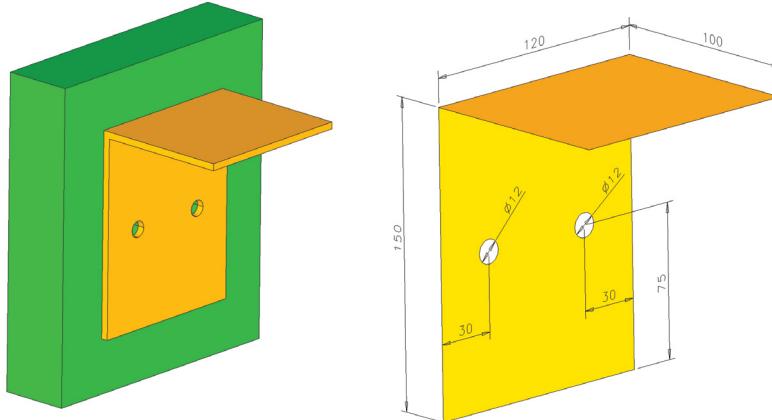


Approach 2) Model at least some portion of the engine and transmission case at the interface (otherwise model both these components completely with a coarse mesh by neglecting small features). Then, represent the other components like the front axle and rear axle using beam elements (approximate the cross sections). Apply constraints at the wheels (not all the dof are fixed but only minimum dofs required to avoid rigid motion or otherwise inertia relief / kinematic dof approach). Please note that the clutch housing, being the critical area, should be meshed fine.

The second option is recommended as the stiffness representation, as well as constraints, are more realistic. The first approach, fixing both sides of the clutch housing, results in over constraining and will lead to safer results (less stress and displacement). Also, it is not possible to consider special load cases like one wheel in ditch, two wheels in ditch, etc.

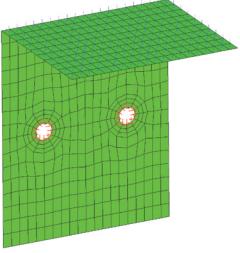
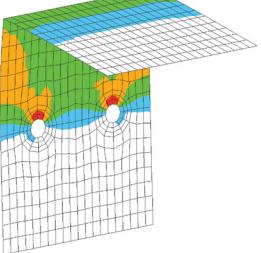
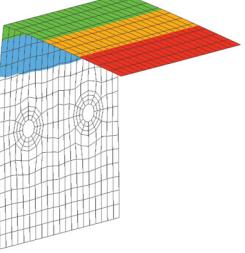
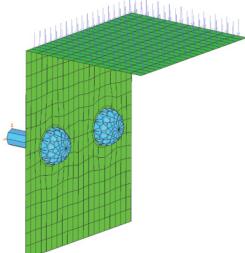
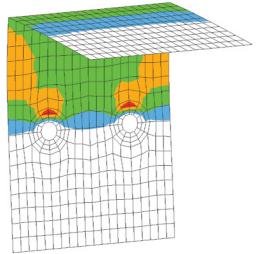
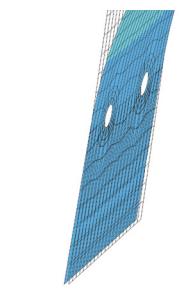
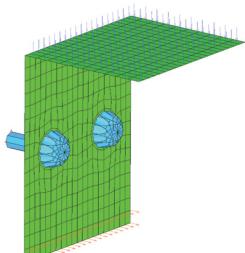
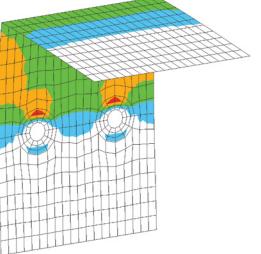
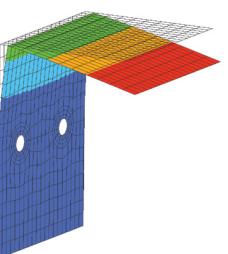
2. Bracket Analysis

Problem definition: A bracket fixed in a rigid wall is subjected to a vertically downward load (180 kg).



If above problem is given to CAE engineers working in different companies then you will find different CAE engineers applying constraints differently:

- Fix the bolt hole edge directly.
- Model the bolt using rigid / beam elements and clamp bolt end.
- Model the bolt, clamp bolt end, bracket bottom edge translations perpendicular to surface fixed.

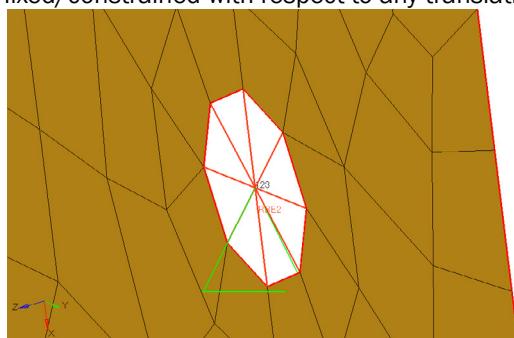
	Stress N/mm ²	Displacement mm
Bolt holes edge fixed	993	15.5
		
Bolt – modeled by beam elements	770	16.2
		
Bolt by beams and Bracket bottom edge only z-dof fixed	758	15.8
		

Applying a direct constraint on the hole edge causes very high stress. The second method shows the deformation of the bracket bottom edge, which is not realistic. Method 3 is recommended. Please observe the difference in the magnitude of the stress and displacement.

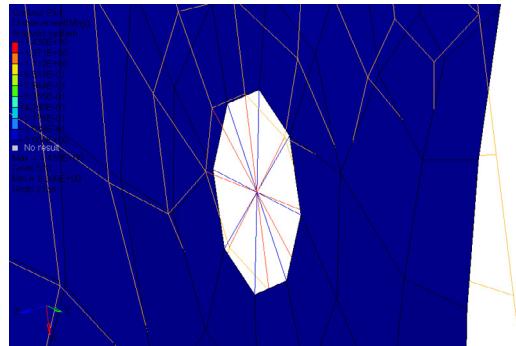
In some organizations it is a standard practice to neglect stress at the washer layer elements (washer area and one more layer surrounding the beam/rigid connection) due to high stresses observed at the beam/rigid and shell/solid connections.

A modified version of the bracket is depicted in the figure below. This time the bracket is pinned to the “wall” by means of 3 simplified screws/bolts. The bolts are represented by rigid elements (RBE2).

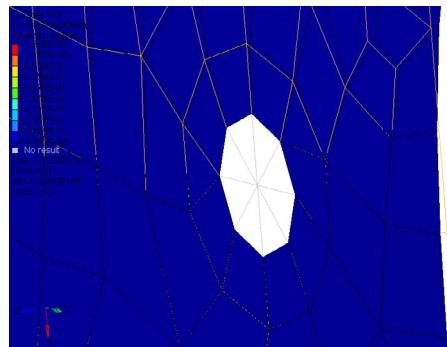
The center of the rigid elements is fixed/constrained with respect to any translational displacements (dof 1-3).



What would you expect to happen? Even though this kind of constraint seems reasonable (i.e. bracket mounted to the wall), the unconstrained rotational dofs allow the central node to rotate. Thus, the hole itself (even though the magnitude is very small) deforms (see figure below).



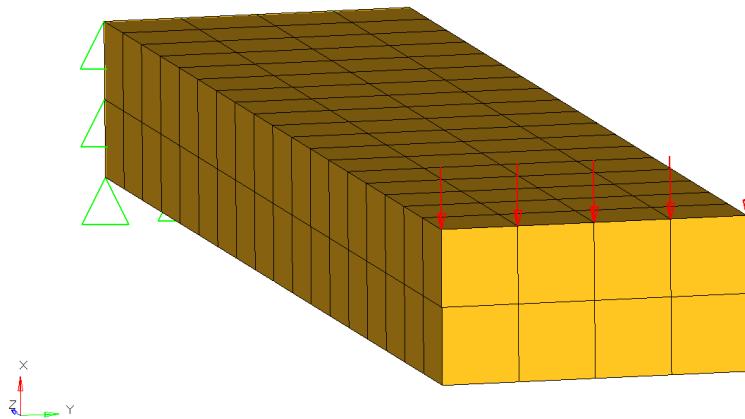
In the figure above, the displacements are scaled by 100. The undeformed shape is plotted in a wireframe mode. Note the deformation of the hole. Is/was this effect really intended?

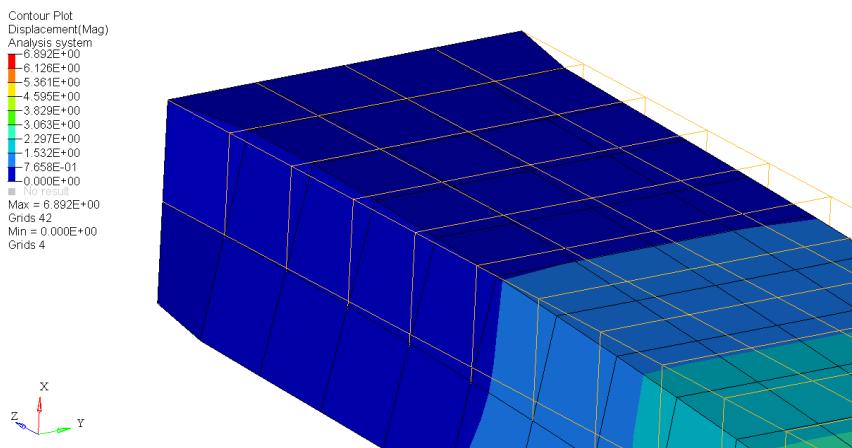


In the figure above, the displacements are scaled by 100. The undeformed shape is plotted in a wireframe mode. All the dofs of the central node are constrained. The hole remains its initial shape and “position”.

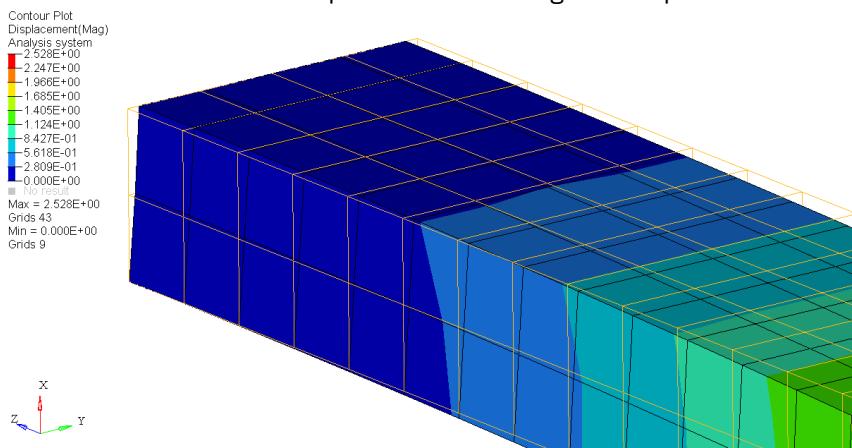
Another illustrative example regarding the effects of the boundary constraints on the modelling results is shown in the figure below. The translational degrees of freedom (dof 123) of the nodes at the rear end of the cantilever are constrained. At the tip a uniform force is applied in negative x-direction.

What would you expect to happen, especially in the vicinity of the constraints?



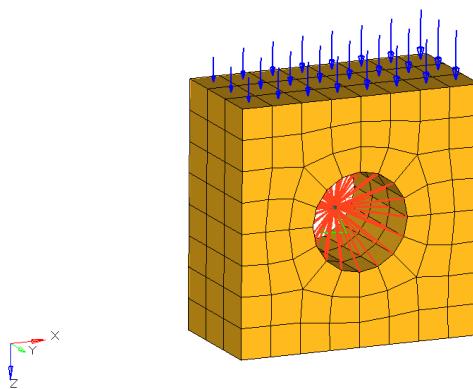


In the figure above, all the nodes at the rear are constrained with respect to any translational displacements. The displacements in the y- and x-direction are scaled by a factor of 200 and 5, respectively. The undeformed mesh is shown as wireframe (orange). Note the thickening at the base of the cantilever accompanied with thinning at the top of the cantilever.



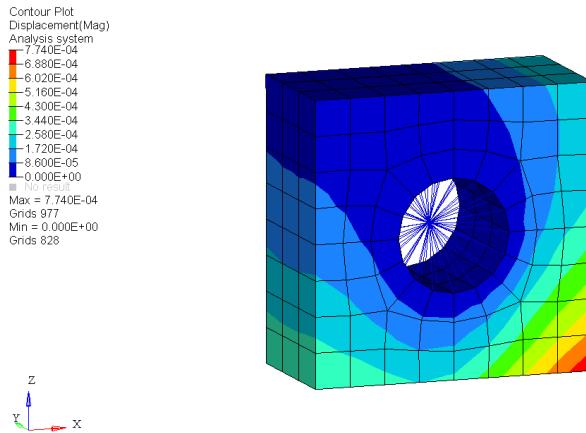
In the figure above, there are modified constraints at the rear of the cantilever. The nodes at the rear part are constraint in the x- and z-direction. In addition, nodes located at the symmetry axis (at the rear) are constraint in the y-direction. The displacements in the y- and x-direction are scale by a factor of 200 and 5, respectively. The undeformed mesh is shown as wireframe (orange). The displacements are quite different compared to the earlier figure. The question to be answered is which model is correct?

Imposing Boundary Conditions (Constraints And Forces) By Means Of RBE2 And RBE3. What Are The Differences?



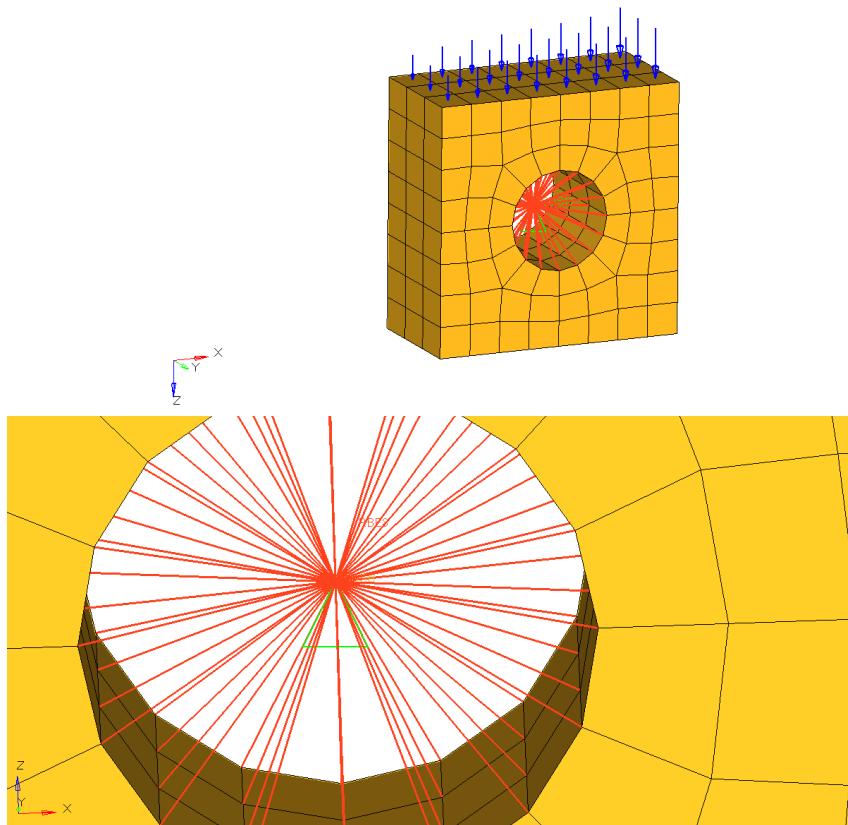
In the figure above, the nodes at the hole are constrained by an RBE2 element where the independent node is fixed with respect to all degrees of freedom (dof123456).

Before looking at the simulation results, ask yourself, what will the displacement contours look like?

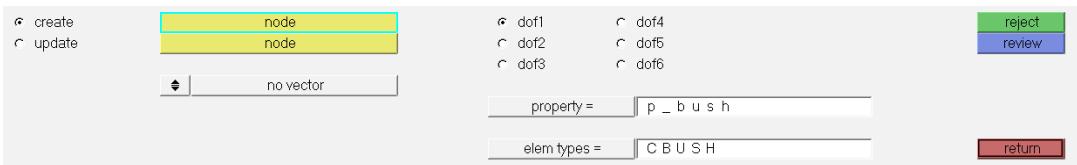


The figure above is a plot of the displacement field. Note that the displacements at the hole are zero. In other words, the RBE2 element “artificially” stiffens the hole.

Next, we are constraining the model by means of a RBE3 element. Note, that the dependent node of the RBE3 element (i.e. the one in the center of the spider) can't be constraint directly. This is because this node would depend on the nodes at the hole and the SPC. A work around is to attach the dependent node to a CBUSH element (with zero length and high stiffness values). The free node of the CBUSH element is then constrained with respect to all degrees of freedom.

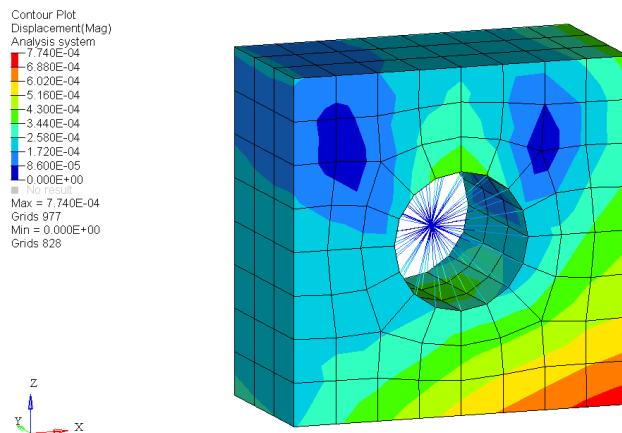
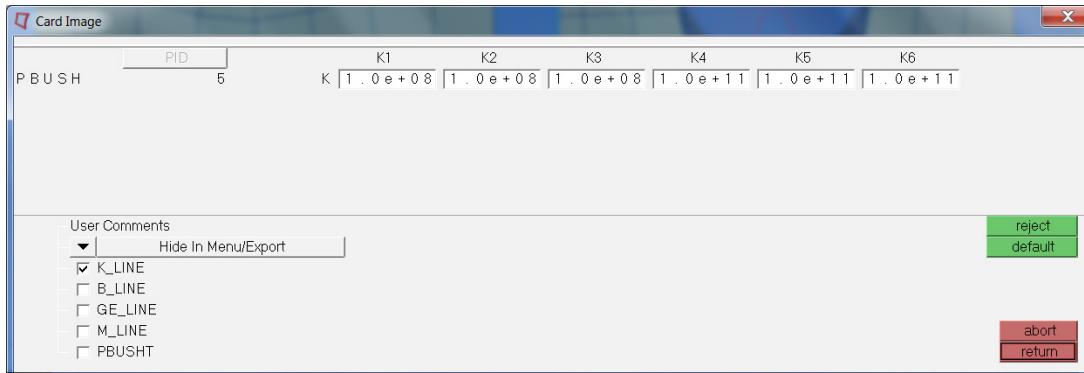


The above figure illustrates how the dependent node of the RBE3 is grounded to an CBUSH element with zero length. All degrees of freedom of the free node of the CBUSH are constrained. CBUSH elements are created in the spring panel which can be accessed by selecting Mesh > Create > 1D Elements > Springs.

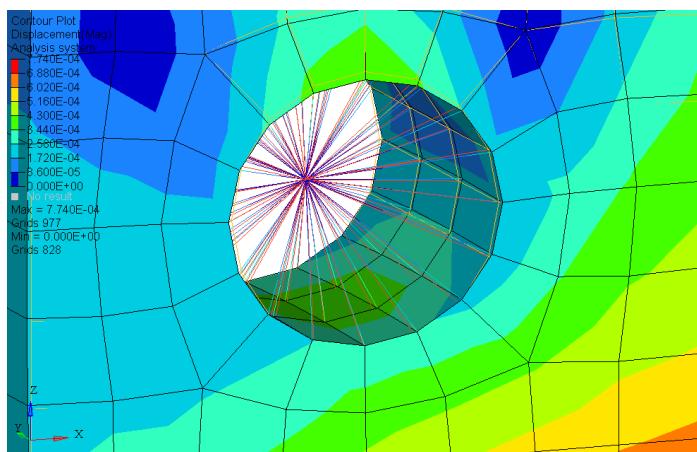


In order to create CBUSH elements, you first need to determine the corresponding “elem types” (default is CELAS), reference a property (can also be assigned later), and select the two end nodes of the spring. The options dof1-6 are irrelevant.

The property definition of the CBUSH element is shown in the following image



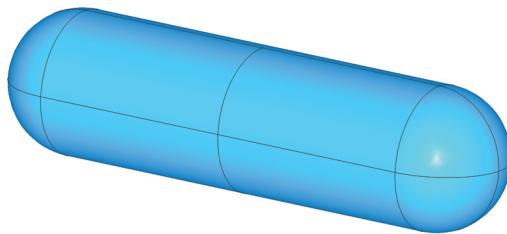
The figure above is a displacement contour plot. The hole is “supported” by a RBE3 element, which in turn is grounded to a CBUSH element with high stiffness values. All degrees of freedom of the free node of the CBUSH element are constrained.



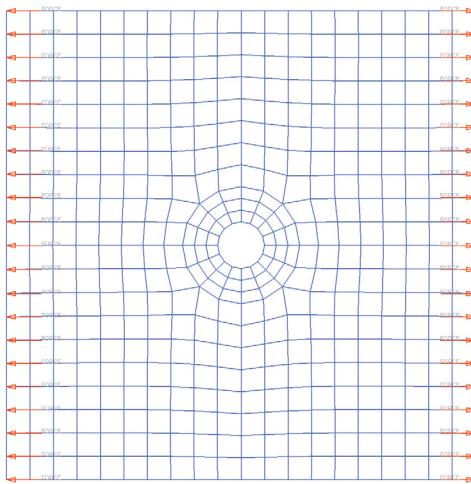
The figure above depicts the displacement contour plot scaled by a factor 200. The hole deforms due to the applied pressure. The RBE3 element thus allows us to model a kind of soft support (whereas the RBE2 approach can be considered as a rigid/stiff support).

3. Pressure Vessel Lying Freely On The Ground And Plate Subjected To Tensile Load On Both The Sides

Sometimes a situation demands for an unconstrained structural analysis like a pressure vessel lying freely on the ground (just placed, no constraint/fixing) or a plate subjected to tensile load on the opposite edges without any constraint. A static analysis problem cannot be solved for an unconstrained structure. It should be fixed at least one node, or alternatively, at a few nodes so as to restrict rigid body motion.



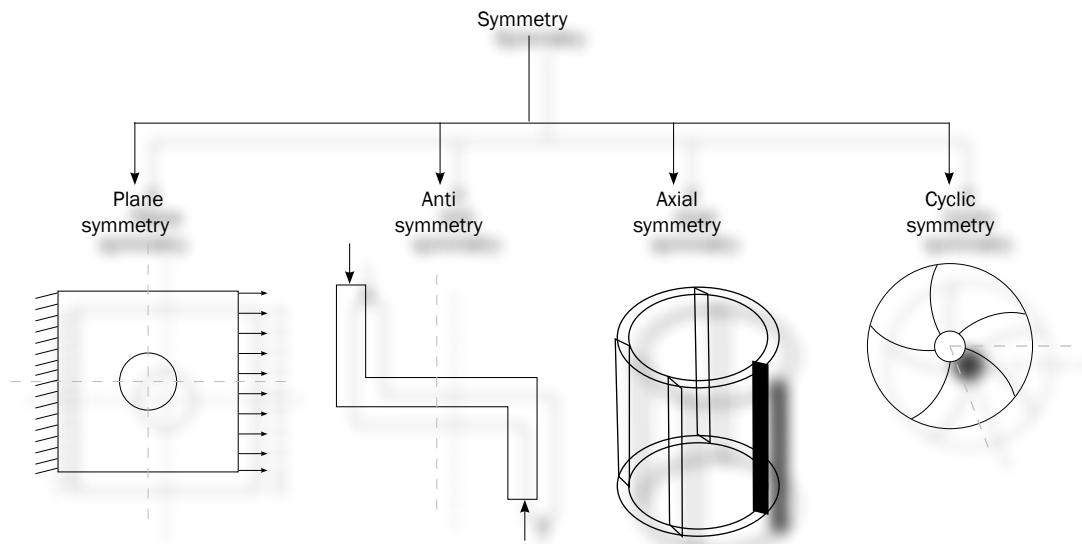
If the problem of the pressure vessel (subjected to internal pressure) or the plate with tensile load is solved without specifying any constraints, then either the solver will quit giving a singularity error message or otherwise report that there is a high stress at an unrealistic location (if the auto singularity option is switched on).



To solve an unconstrained structural problems there are 2 ways:

- 1) Approximate approach: Create spring / beam elements (negligible stiffness value) on the entire circumference (outer edges or surface nodes) and apply the constraint at the free end of spring or beam.
- 2) Recommended: Inertia relief method or defining kinematic dofs in the model (see HyperWorks help documentation and corresponding tutorials)

15.3 Symmetry



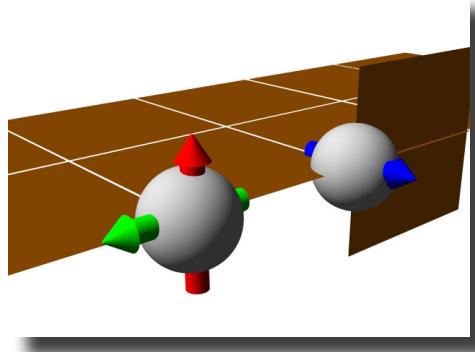
Condition For Using Any Type Of Symmetry

Symmetric conditions could be used only when both the following conditions are fulfilled.

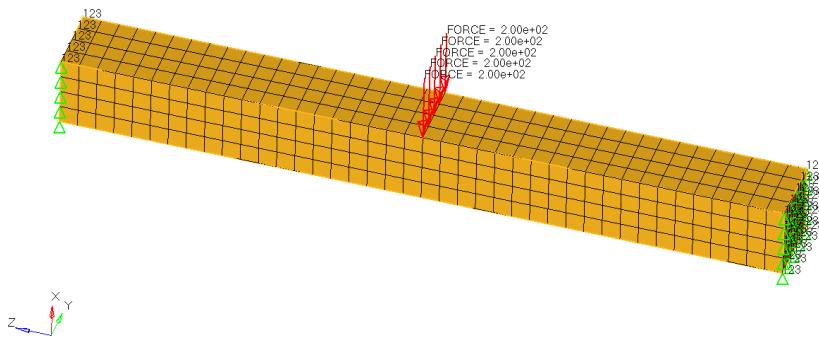
- 1) Geometry is symmetric
- 2) Boundary conditions (forces and constraints) are symmetric.

Advantages: Half, quarter or a portion of the model could be used for analysis, resulting in fewer dofs and computational cost.

Which dof must be constrained at the symmetry level?

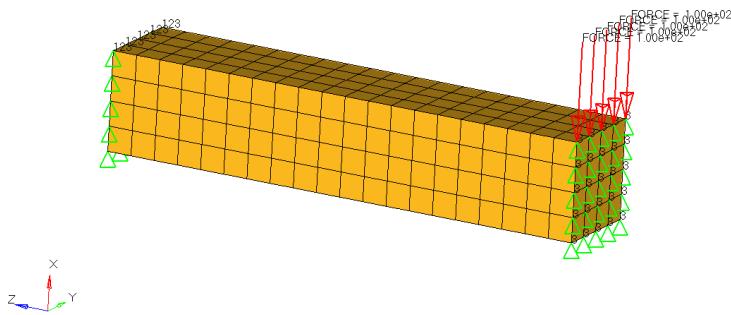


In the figure above, the dark vertically oriented plane represents the plane of symmetry. The finite elements nodes are colored gray, whereas possible nodal rotations are shown by means of blue, green and red arrows. Nodal rotations with respect to the green and red axis/arrows would “move/rotate” the node out of the plane of symmetry (just imagine the arrows would be glued to the nodes). Hence these degree of freedoms (dof) must be constrained. In contrast, nodal rotations with respect to the blue axis/arrow are not needed to be constrained. As the nodes of solid elements do only allow translational displacements, one just need to constrain any out of symmetry plane motions.

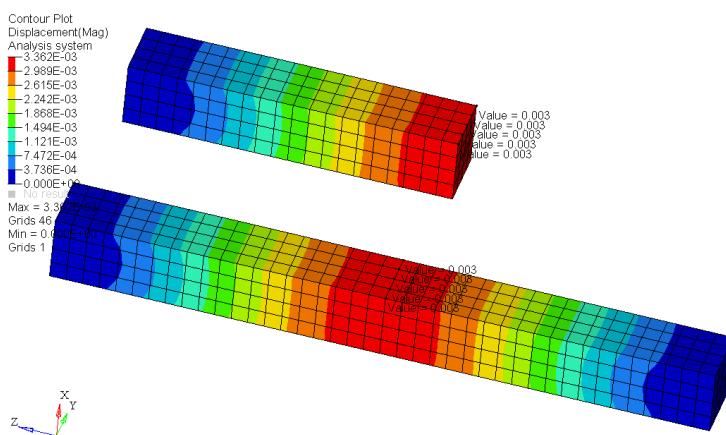


In the figure above the full model is considered. The beam ends are constrained with respect to any translational displacements (dof 123). A vertical load of 200 N is applied at its center.

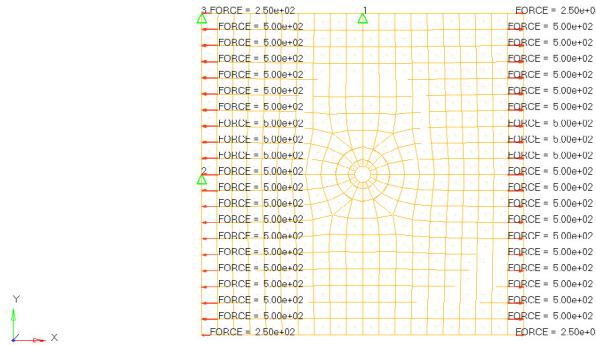
If the symmetry plane is in the x-y plane then the translational displacements in its normal direction i.e. z-direction (dof 3) need to be constrained. On the other hand, we don't need to fix/delete rotations with respect to z-axis as solid elements do not allow nodal rotations. Remember, the nodes at the symmetry plane are not allowed to move (or rotate) out of the plane of symmetry.



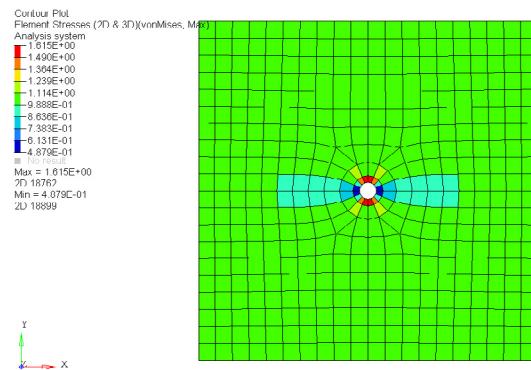
In the image above only a half model is considered. At the plane of symmetry, the z-displacements (dof 3) are constrained. In addition, the original force is divided by two (as it acts only on half of the structure).



Let us consider a symmetrical plate with a hole subjected to symmetrical loads on the two opposing edges.

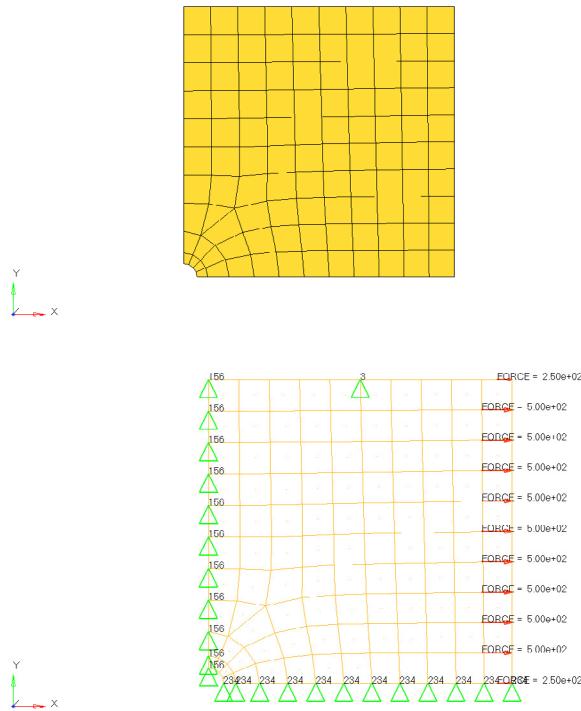


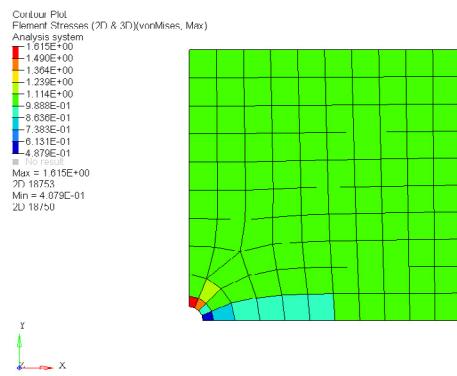
Above the full plate model is shown (and serves as a reference model)



Contour plot of element stresses (von Mises).

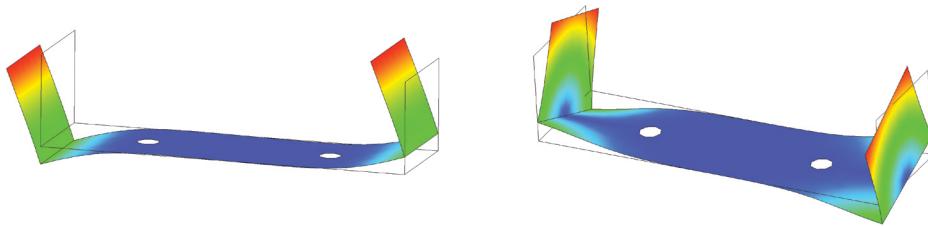
In the next step, a quarter segment of the plate is investigated. The corresponding loads and constraints are shown in the figure below.



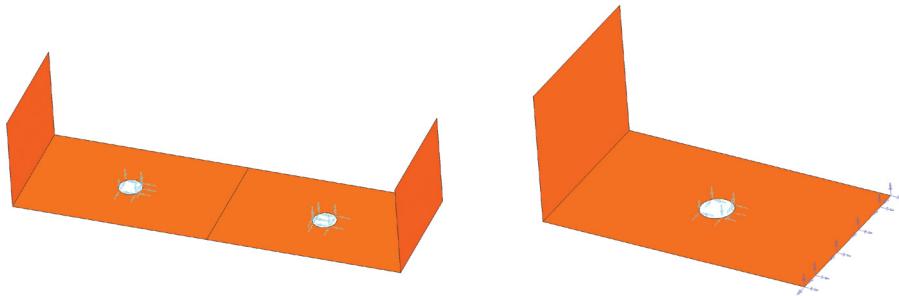


Contour plot of the element stresses (von Mises).

Limitations: Symmetric boundary conditions should not be used for dynamic analysis (natural frequency and modal superposition solver). A symmetric model (half or portion of part) would miss some of the modes (anti nodes or out of phase modes) as shown below:



Natural Frequency comparison for Full and Half symmetric model



Mode No.	Hz	
	Full model	Half symmetric model
1	448	448
2	449	718
3	718	1206
4	719	2078
5	1206	2672
6	1211	4397

Question: We need to do a simulation of a casting component to decide the best option out of two casting materials. What will be the differentiating parameters, since the modulus of elasticity and Poisson's ratio for both of the materials are the same. Secondly, we do not have stress strain data for both of the materials.

1) If the nature of the loading is static (most of the time static and rarely dynamic) and the company has only a linear static solver:

Different grades of cast iron have different ultimate strength (or proportional yield strength). Linear static stress is independent of material and both the materials would report the same stress. The decision could be made based on the ultimate strength (and endurance strength). Say the maximum stress as per the FEA is 300 N/mm². For material 1, the ultimate stress is 350 and for material 2 it is 500 N/mm². In this situation, the second material is clearly the choice. Consider another case. Suppose the maximum stress reported by the FEA is 90. One can calculate the approximate value of endurance strength. For grey cast iron, the endurance strength = 0.3 * ultimate strength. Therefore, for material 1 the endurance strength is 105 N/mm² and should be preferred over the second one.

2) If the nature of the loading is dynamic and subjected to a sever load: Fatigue analysis is strongly recommended. Commercial fatigue analysis software usually provide material library. Specifying the appropriate material grade, surface finish, etc. and calculations for the life / endurance factor of safety would lead to realistic and optimum selection out of the two options.

15.4 Creating Loadsteps In HyperMesh

And finally, once the constraints and loads are specified, a corresponding loadstep needs to be created (otherwise the Finite Element program would not know what to do with these entities). In HyperMesh, this is done by selecting Setup > Create > LoadSteps.



The image above shows the Load Step panel in HyperMesh. This panel is used to define a loadstep. First, specify the name and the type of the loadstep. The load collector with the constraints are referenced using SPC and the forces/pressure are referenced using LOAD.

The loadstep will then be listed in the Model Browser along with all the other model entities.

Now the model can be exported and the analysis can be started!

15.5 Discussion On AUTOSPC In OptiStruct

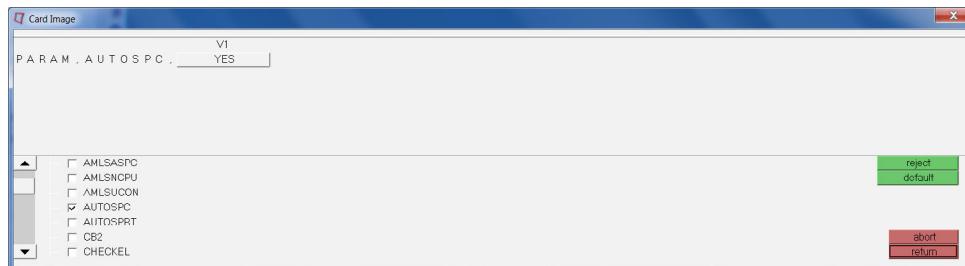
This section is based on discussions with Kristian Holm, Juergen Kranzeder and Bernhard Wiedemann. Written by Matthias Goelke

A First Look At AUTOSPC

AUTOSPC checks the global stiffness matrix for degrees-of-freedom with no stiffness. If found, these degrees-of-freedom are automatically constrained. In the *.out file, the nodes which are virtually constrained are listed.

Note, that AUTOSPC,YES is the default setting.

AUTOSPC is defined under Control Cards > PARAM > AUTOSPC > YES/NO.



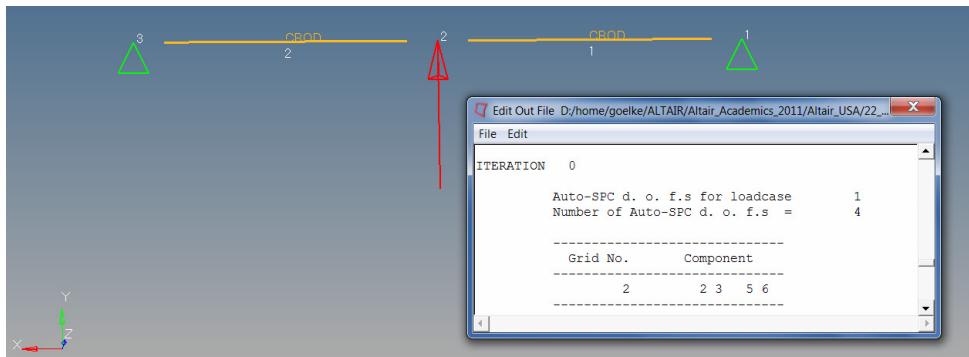
Example 1

AUTOSPC,YES

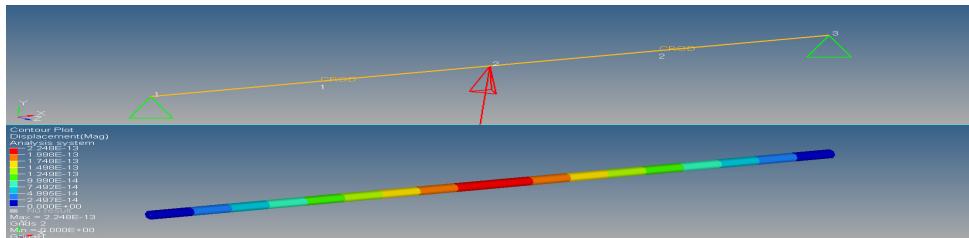
In this example two CROD elements, which support tension, compression and torsion, are subjected to a vertical load at their common node.



Running the analysis yields the following information:



At node 2 the dofs in 2 and 3, (2=y, 3=z translation), 5 and 6 (5= rotation y axis, 6= rotation z-axis) are not available. With the setting AUTOSPC =YES, these unsupported dofs are automatically constrained. As a consequence, the analysis gets completed without warning or error messages.



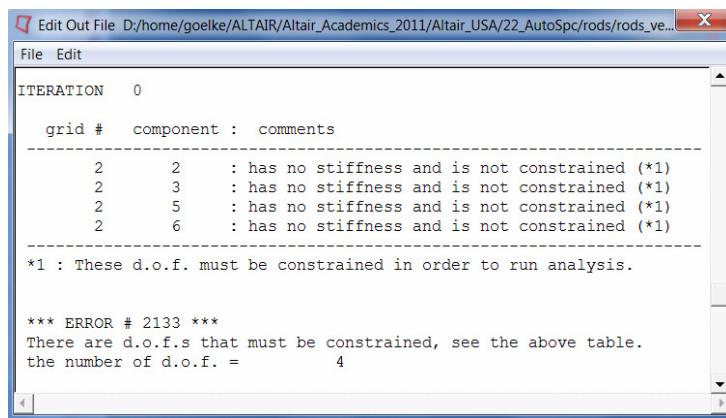
However, note that the displacements are in the order of $10e^{-13}$ mm, which is "zero" (despite the applied loading).

In a first reaction, this result may provoke the idea that material attributes, cross-sectional properties of the rods, or the magnitude of the load is not properly defined. However, if the focus of the analysis is about bending (which seems to be the case due to the applied loads), the chosen element type is simply wrong. Instead of CROD elements (which only allow for compression, tension and torsion), CBAR or CBEAM (capable of bending) should be applied.

Deactivating AUTOSPC (=NO) demonstrates that the model setup is wrong:

AUTOSPC, NO

Running the same model but with AUTOSPC set to NO, the analysis stops with the error message depicted below:



```

Edit Out File D:/home/goelke/ALTAIR/Altair_Academics_2011/Altair_USA/22_AutoSpc/rods/rods_ve...
File Edit
ITERATION 0
grid # component : comments
-----
2 2 : has no stiffness and is not constrained (*1)
2 3 : has no stiffness and is not constrained (*1)
2 5 : has no stiffness and is not constrained (*1)
2 6 : has no stiffness and is not constrained (*1)
-----
*1 : These d.o.f. must be constrained in order to run analysis.

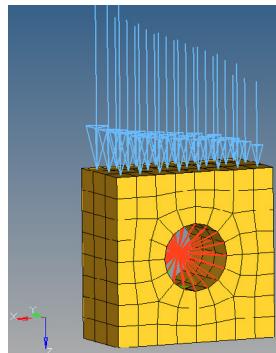
*** ERROR # 2133 ***
There are d.o.f.s that must be constrained, see the above table.
the number of d.o.f. = 4

```

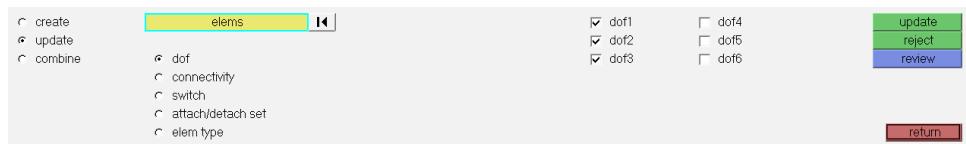
This time, as AUTOPC is set to be NO, grid 2 has no stiffness regarding dof 2, 3, 5, and 6. The analysis stops, requesting that these dofs must be constrained in order to run analysis.

Example 2

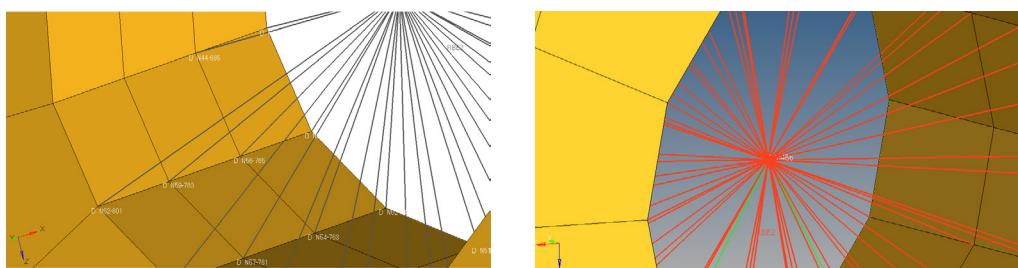
AUTOSPC, YES



The solid mesh (first order hexahedral elements) is constrained (all dofs are fixed) at the independent node of the RBE2 element.



- The independent node of the RBE2 element is pinned/fixed with respect to all dofs (1-6).
- All dofs of the RBE2 element are active.
- The nodes of the hexahedral element (with dof 1-3; no rotation) are coupled to the dependent nodes of the RBE2 element (with dof 1-6).



Will this cause any problems? What do you expect?

The *.out file provides the following information:

```
*** INFORMATION # 742
The dependent rotational d.o.f. of this rigid element is removed.
  RBE2 element id =      614
  independent grid id =    1159
  dependent grid id =     498
This is because there is no need to constrain the rotational d.o.f. of
any of the dependent grids.

*** INFORMATION # 743
The total number of rigid elements, whose rotational dependent d.o.f.
are removed because there is no need to constrain those d.o.f., is      1
```

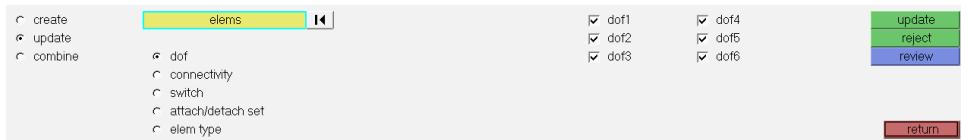
Since the solid elements do not support dof 4-6, the rotational dofs of the RBE2 element (dof 4-6) have been removed.

However, during the analysis with AUTOSPC,YES the solver detects nodes (altogether 62 nodes) whose rotational dofs have been removed. Now, AUTOSPC will automatically constraints these nodes.

```
ANALYSIS RESULTS :
-----
ITERATION   0
Auto-SPC d. o. f.s for loadcase      1
Number of Auto-SPC d. o. f.s =      186
-----
Grid No.      Component
-----
  457          4 5 6
  458          4 5 6
  459          4 5 6
  460          4 5 6
  461          4 5 6
  497          4 5 6
  498          4 5 6
  499          4 5 6
  500          4 5 6
  501          4 5 6
  523          4 5 6
  524          4 5 6
```

Altogether 62 nodes are listed with a total of 186 dofs

Note, that by default, the RBE2 element allows for translation and rotation:



Simply deactivate dof 4, 5, and 6 to avoid the automatic pinning/fixing of nodes due to AUTOSPC, YES.

AUTOSPC, NO

Running the same model with AUTOSPC,NO the following information is listed in the *.out file:

```
1 PARAM(s) were set:
AUTOSPC ,NO

NOTE # 1929
Created RAM scratch area: total size 30 MB.

*** INFORMATION # 741
No need to constrain the rotational d.o.f. of this dependent grid.
  RBE2 element id =      614
  independent grid id =    1159
  dependent grid id =     498
This is because there isn't any stiffness and load on the rotational
d.o.f. of the dependent grid.

*** INFORMATION # 741
No need to constrain the rotational d.o.f. of this dependent grid.
  RBE2 element id =      614
  independent grid id =    1159
  dependent grid id =     500
This is because there isn't any stiffness and load on the rotational
d.o.f. of the dependent grid.

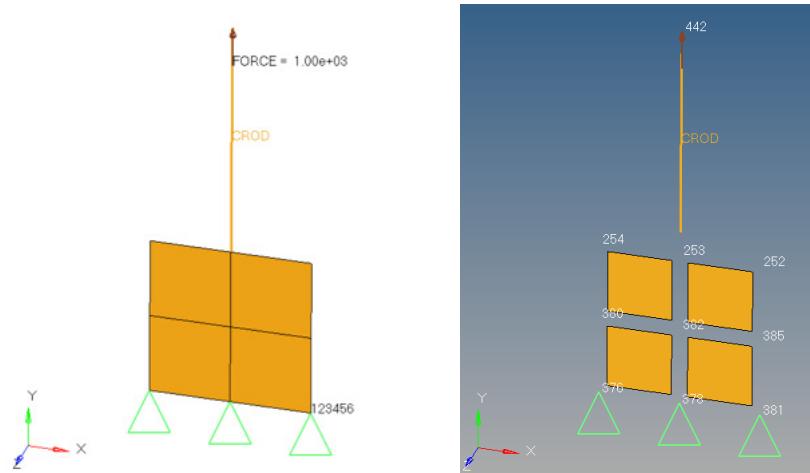
*** INFORMATION # 741
No need to constrain the rotational d.o.f. of this dependent grid.
  RBE2 element id =      614
  independent grid id =    1159
  dependent grid id =     499
This is because there isn't any stiffness and load on the rotational
d.o.f. of the dependent grid.

*** INFORMATION # 743
The total number of grids whose rotational d.o.f.s need not be
constrained in RBE2 elements is      62
```

Again we are informed that the rotational dofs of 62 nodes (=186 dofs) need not to be constrained.

At the end the results of Example 2 with AUTOSPC,YES or NO are identical.

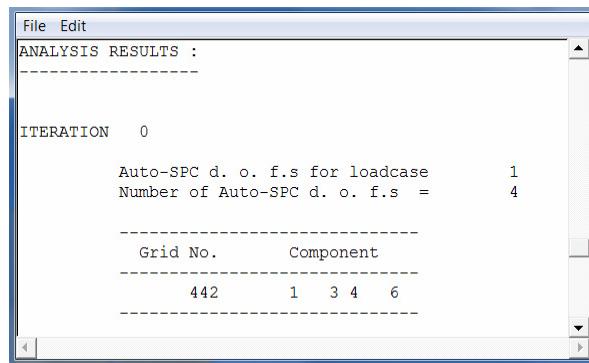
Example 3



In this model, a ROD element that supports compression, tension and torsion (here dof 2, dof 5) is attached to shell elements (with dof 1-6) and subjected to a force in positive y-direction.

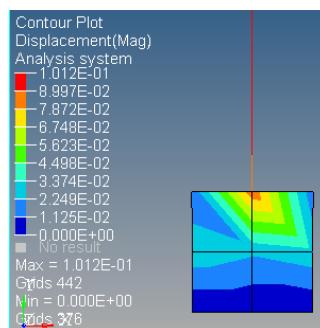
- Coupling of Rod element (dof2 and dof5) with shell element (dof1-6)
- No constraints at the free end of the ROD element (location of the applied force)

Do you see any problems? What do you expect?



Since the dofs 1, 3, 4, and 6 are not available/supported at node 442, AUTOSPC, YES places “virtual” constrains on this node.

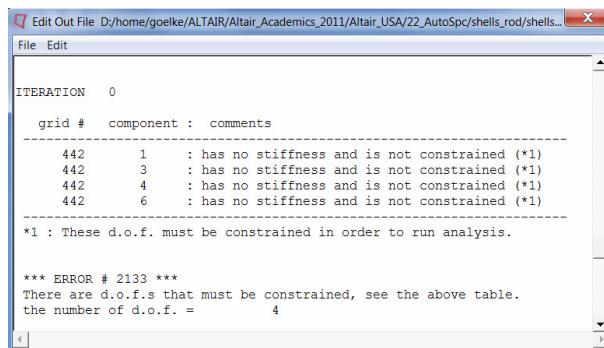
As a consequence, we do get results – but can we trust the results?



This question becomes even more striking by looking at the model with the setting AUTOPC, NO.

AUTOSPC, NO

The solver “detects” that node 442 is missing any stiffness associated with dof 1, 3, 4, and 6 (due to the incompatibility of dofs at node 253). Since we do not allow the solver to add constraints to this node, the analysis fails:



The screenshot shows a software window titled "Edit Out File D:/home/goelke/ALTAIR/Altair_Academics_2011/Altair_USA/22_AutoSpc/shells_rod/shells...". The window displays the following text:

```
ITERATION 0
grid # component : comments
-----
442      1    : has no stiffness and is not constrained (*1)
442      3    : has no stiffness and is not constrained (*1)
442      4    : has no stiffness and is not constrained (*1)
442      6    : has no stiffness and is not constrained (*1)
-----
*1 : These d.o.f. must be constrained in order to run analysis.

*** ERROR # 2133 ***
There are d.o.f.s that must be constrained, see the above table.
the number of d.o.f. =        4
```

As reported in the *.out file, node 442 has no stiffness for dof 1, dof3, dof4, and dof 6, causing the analysis to stop with a fatal error!

So in the first scenario we do get results because of an introduced stiffness related to AUTOSPC,YES at node 442. In the latter with AUTOSPC,NO the same model cannot be run at all. In order to overcome the model error we either need to constrain node 442 according to the information provided in the table – or we make use of a CBAR/CBEAM element (with bending stiffness) instead of CROD.

Remark

AUTOSPC,YES is the default setting.

Though the solver run may be completed successfully (because of AUTOSPC,YES), it doesn't necessarily mean that the model is physically correct. User defined SPC's may be incomplete or incorrect (not always that obvious at all) – AUTOSPC,YES may then automatically add further constraints which may lead to intriguing results as depicted by the examples shown above.

Make sure and always check the *.out file for additional information. Also, you may run the model again, this time applying the dofs (constraints) as listed in the *.out file and AUTOSPC,NO.

15.6 Recommended Tutorials And Videos

Recommended Tutorials:

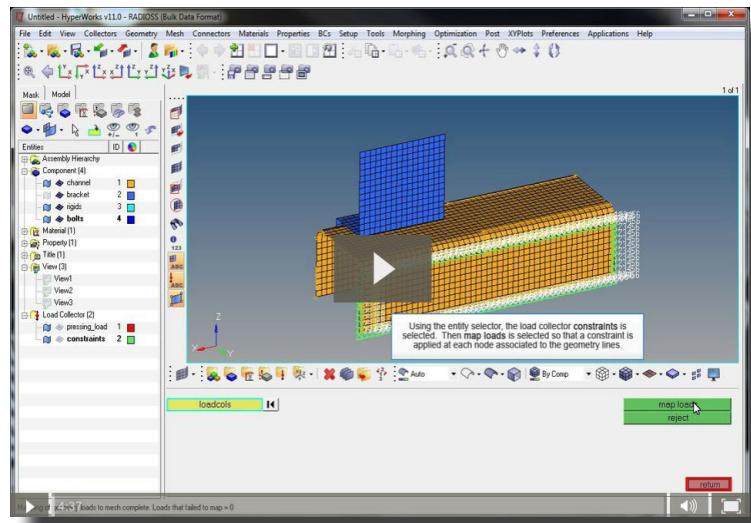
The following tutorials are included in your HyperWorks installation (see HyperWorks help documentation).

- HM-4000: Setting up Loading Conditions
- HM-4040: Working with Loads on Geometry

Recommended Videos

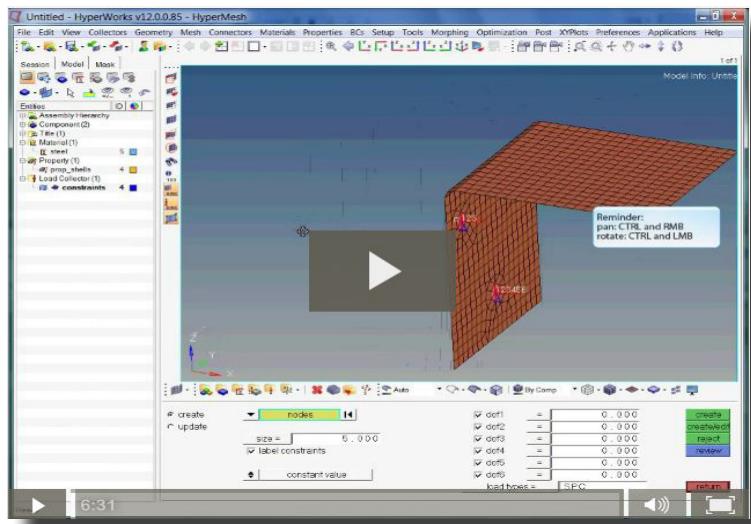
Creating boundary conditions

(<http://altair-2.wistia.com/medias/3m5mzg42dn>)



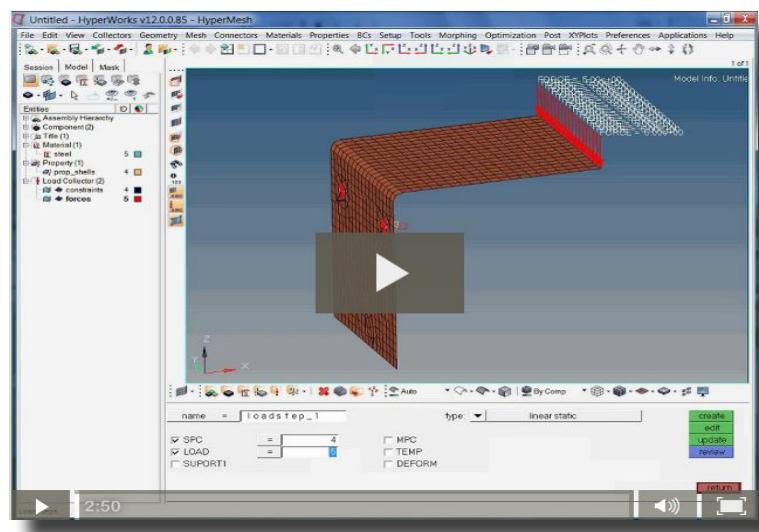
Loads and constraints

(<http://altair-2.wistia.com/medias/tq0gr7zafo>)



Loadstep definition

(<http://altair-2.wistia.com/medias/52c93wwc5x>)



Where Do You Get Your Loads To Assess Component Strength And Durability

(<http://altair-2.wistia.com/medias/91h98xs0h9>)

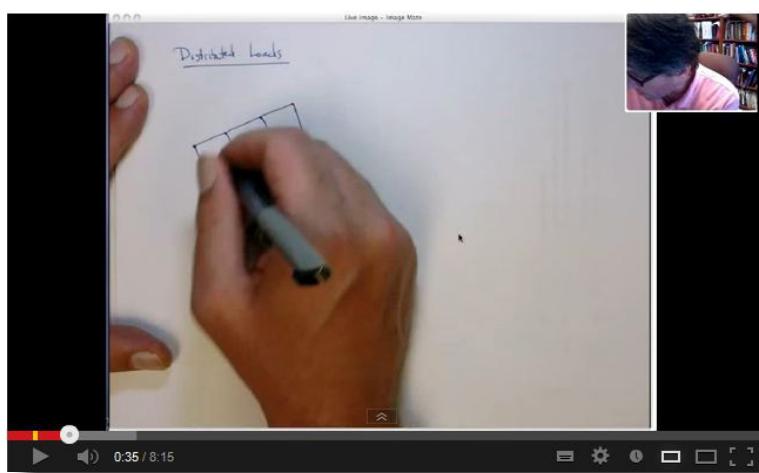


More General (not HyperWorks related)

Distributed loads in finite elements, part 1

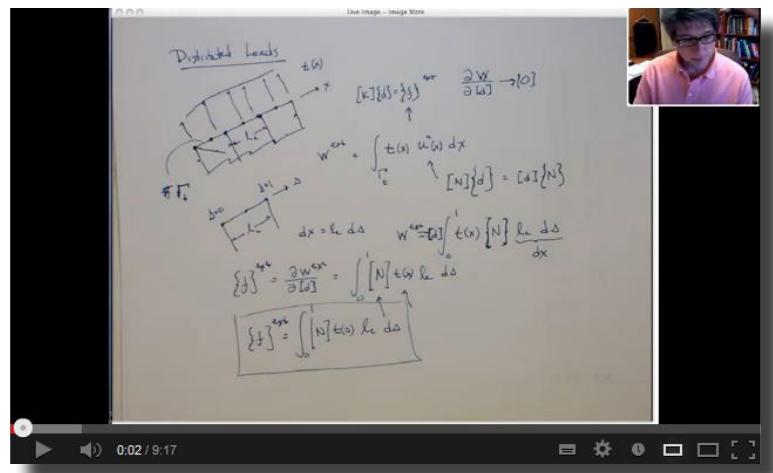
(Prof. J. Chessa; University Texas;

<http://youtu.be/8lwJQmDo7zQ>)



Distributed loads in finite elements, part 2

(Prof. J. Chessa; University Texas; <http://youtu.be/gtBFrF-7ntOs>)



Recommended Tips And Tricks

- Boundary Conditions – Loads Summary Tool
- Beam - Pressure Loading

Presentation (PDF document)

Where are your loads coming from - Suspension and Chassis Loads

(http://www.altairuniversity.com/wp-content/uploads/2013/07/Loads_Extraction_120309.pdf)



15.7 Student Racing Car Project - Boundary Conditions

BC's & Loads

Three different loadsteps are considered:

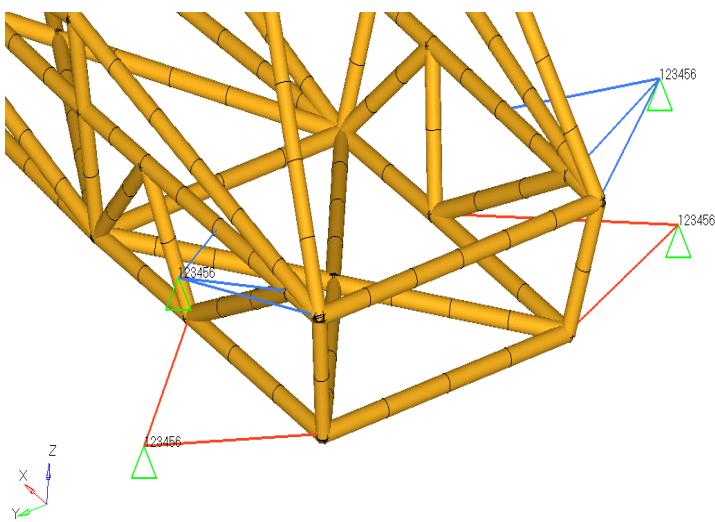
- Torsion
- Bending
- Shear

In all three loadsteps the frame is constraint in the same way (see image below). Thus, 4 loadcollectors (no Card Images needed) must be created.

Entities	ID	
Assembly Hierarchy		
Beam Section (1)		
Card (3)		
Component (4)		
Load Collector (4)		
Constraint	1	■
Shear	2	■
Bending	3	■
Torsion	4	■
Load Step (3)		
Material (1)		
Property (1)		
25x2	2	■
Title (1)		
Model Info	1	□

Constraints

The corresponding nodes are fixed with respect to translational and rotational displacements (dof 123456). To create constraints, from the menu bar select BCs > Create > Constraints.



Torsion

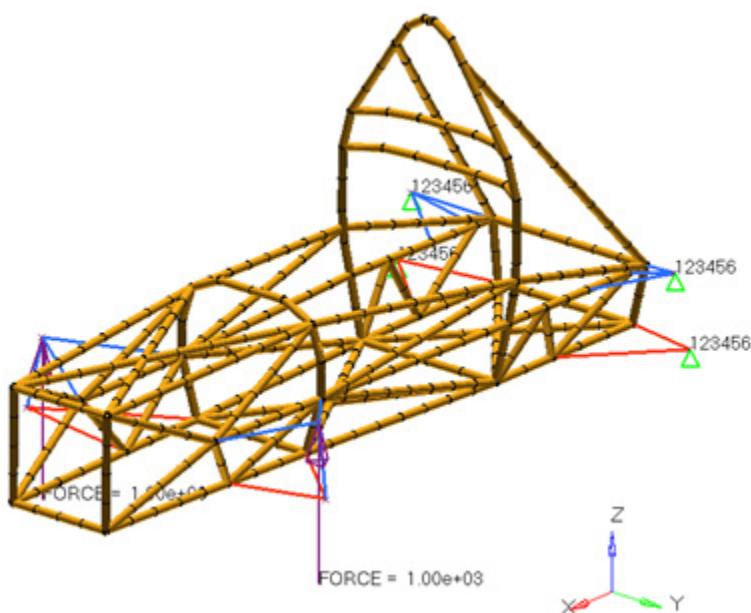
Torsion is imposed through vertically applied forces. To create forces, from the menu bar select BCs > Create > Forces. Of course the referenced forces and magnitudes are just "dummy" assumptions. The frame is constrained as described before.

The corresponding torsional loadstep is defined in the "LoadStep" panel (Setup > Create > LoadSteps).

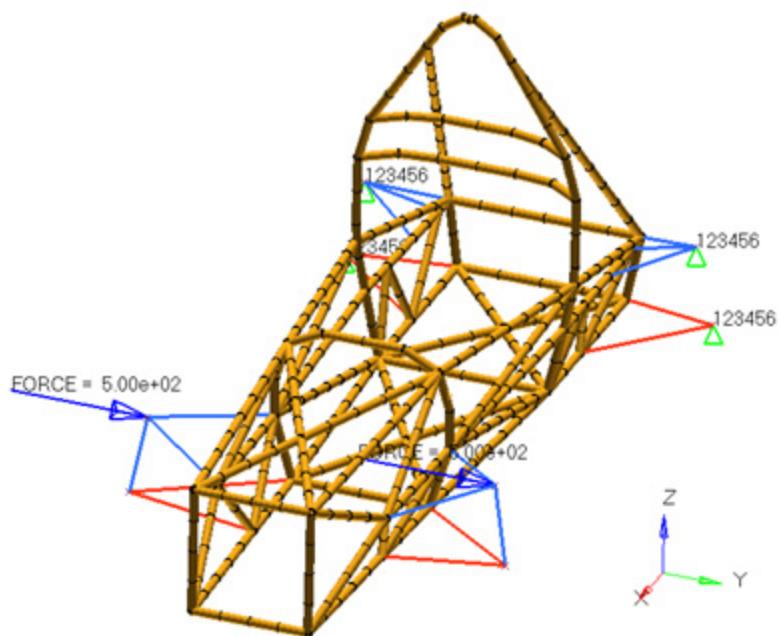


Bending & Shear

The working steps related to the setup of the bending and shear load cases are pretty much the same. The orientation of the bending and shear forces is shown in the images below. Care must be taken, that these forces are stored in their respective load collectors (no Card Image needed). The loadstep named "Bending" then references the loadcollectors "Bending" and "Constraint"; the loadstep "Shear" the loadcollectors "Shear" and "Constraint".



Load case: Bending



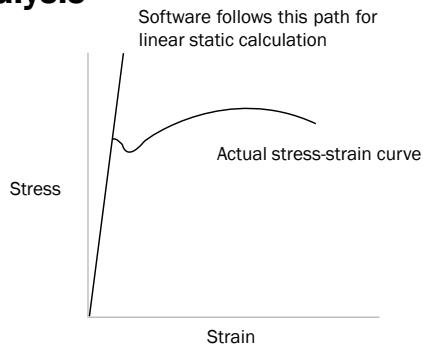
Load case :Shear

This completes the model set-up. The model is saved as a binary *.hm file (including CAD information) and then exported as an OptiStruct analysis deck (ASCII format).

16 Linear Static Analysis

This chapter includes material from the book “**Practical Finite Element Analysis**”. It also has been reviewed and has additional material added by Debdatta Sen and Matthias Goelke.

16.1 Linear Static Analysis



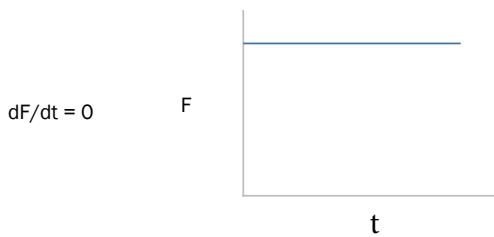
Linear

Linear indicates a linear elastic behaviour of the material. That is, the linear portion of the Stress Strain Curve a straight line following Hooke's Law : $\sigma = E \epsilon$ which can be interpreted as the equation of a straight line ($y = m x$) passing through the origin. “E”, the Elastic Modulus, is the slope of the curve and is a constant. In real life, after crossing the yield point, the material follows a non-linear curve, but the solver follows the same straight line. Components are broken into two separate pieces after crossing the ultimate stress, although a linear static analysis never shows failure in this fashion. It shows a single unbroken part with high stresses at the location of failure. Very large, unrealistic deformations may also be seen. An analyst has to conclude whether the component is safe or if it failed by comparing the maximum stress value with the yield or ultimate stress. Hence it's necessary for the analyst to decide, if under the given loading conditions, a Linear Static Analysis can be carried out.

Static

There are two conditions for static analysis:

- 1) Force is static meaning there is no variation with respect to time (dead weight)



2. Equilibrium condition - $\sum \text{Force} = 0$, $\sum \text{Moments} = 0$

$$\begin{array}{ll} \sum F_x = 0 & \sum M_x = 0 \\ \sum F_y = 0 & \sum M_y = 0 \\ \sum F_z = 0 & \sum M_z = 0 \end{array}$$

The FE model should fulfil this condition at each and every node. The summation of all external forces and moments should at all locations, be equal to the reaction forces and moments.

Practical Applications: This is the most commonly used analysis. All Aerospace, Automobile, Offshore and Civil engineering industries perform linear static analysis.

16.2 Linear Static Analysis Example Using HyperMesh

Theoretical Result

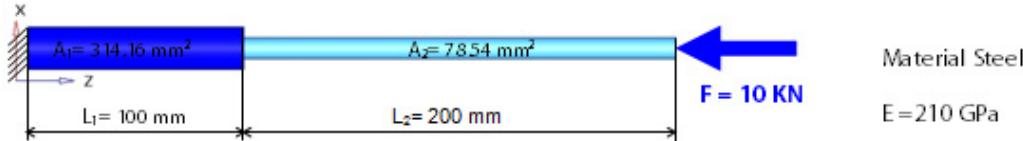
In mechanics we can define static state as the state of a system that is in equilibrium under an action of balanced forces and torques so that they remain at rest ($V=0$). A static load is one which does not vary. If it changes slowly, the structure's response may be determined with static analysis, but if it varies quickly (relative to the structure's ability to respond), the response must be determined with a dynamic analysis. When solving a static problem, all finite element solvers will solve the following equation:

$$\mathbf{K}\mathbf{x} = \mathbf{f}$$

Where:

- **K** : It is the global stiffness matrix
- **x** : It is the displacement vector response to be determined.
- **f** : It is the external forces vector applied to the structure.

We will now consider a small static example that will be used to better understand what a static analysis is. The example is showed on the following image:



Simple rod example

This is a stepped column consisting of two uniform sections with different cross-sectional areas.

The methodology that will be described here can be easily extended to any kind of problem. First we need to represent the structure as nodes and elements. For this case, it is clear that we need at least 2 elements, one for each distinct section.

Let's consider the following finite element model to solve this linear static problem. In our case, we will use rod elements as showed on the following image:



The model comprises of 3 nodes {1, 2, and 3} and 2 elements {1 and 2}. Consider a material that is associated to a rod element with the cross-section A and length L. Every node is assumed to have only one DOF (x) and the stiffness matrix for this one-dimensional rod element can be written as:

$$\mathbf{K} = \begin{bmatrix} \frac{AE}{L} & -\frac{AE}{L} \\ -\frac{AE}{L} & \frac{AE}{L} \end{bmatrix}$$

We can then evaluate the matrix for each element based on the input data:

$$\mathbf{K}_1 = \begin{bmatrix} \frac{314.16 * 210}{100} & -\frac{314.16 * 210}{100} \\ -\frac{314.16 * 210}{100} & \frac{314.16 * 210}{100} \end{bmatrix} = \begin{bmatrix} 659.74 & -659.74 \\ -659.74 & 659.74 \end{bmatrix}$$

$$\mathbf{K}_2 = \begin{bmatrix} \frac{78.54 * 210}{200} & -\frac{78.54 * 210}{200} \\ -\frac{78.54 * 210}{200} & \frac{78.54 * 210}{200} \end{bmatrix} = \begin{bmatrix} 82.47 & -82.47 \\ -82.47 & 82.47 \end{bmatrix}$$

Now the next step is to assemble these two element stiffness matrices to form the global stiffness matrix:

$$\mathbf{K}_G = \begin{bmatrix} 1 & 2 & 3 \\ 1 & 659.74 & -659.74 & 0 \\ 2 & -659.74 & 659.74 + 82.47 & -82.47 \\ 3 & 0 & -82.47 & 82.47 \end{bmatrix}$$

Now we need to define the vector matrices for forces and the displacements:

$$\mathbf{f} = \begin{Bmatrix} 0 \\ 0 \\ -10 \end{Bmatrix} \quad \mathbf{x} = \begin{Bmatrix} x_1 \\ x_2 \\ x_3 \end{Bmatrix}$$

Finally, the global system can now be represented in terms of the global stiffness matrix, displacement and force matrices as:

$$\begin{bmatrix} 659.74 & -659.74 & 0 \\ -659.74 & 742.21 & -82.47 \\ 0 & -82.47 & 82.47 \end{bmatrix} \begin{Bmatrix} x_1 \\ x_2 \\ x_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ -10 \end{Bmatrix}$$

Node 1 is fixed in this model and hence the displacement for this node would be zero. So we can eliminate the first line and the first column of our system:

$$\begin{bmatrix} 742.21 & -82.47 \\ -82.47 & 82.47 \end{bmatrix} \begin{Bmatrix} x_2 \\ x_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -10 \end{Bmatrix}$$

To solve this problem we just need to invert the global stiffness matrix and multiply it to both sides:

$$\begin{Bmatrix} x_2 \\ x_3 \end{Bmatrix} = \begin{Bmatrix} -0.0152 \\ -0.1364 \end{Bmatrix}$$

This gives us the displacement values at nodes 2 and 3. With the displacement vector defined, it is now possible to determine the element strain, stresses and forces:

STRAIN

$$\varepsilon_1 = \frac{\Delta L}{L} = \frac{x_2 - x_1}{L_1} = \frac{-0.0152 - 0}{100} = -1.52 * 10^{-4} \text{ mm/mm}$$

$$\varepsilon_2 = \frac{\Delta L}{L} = \frac{x_3 - x_2}{L_2} = \frac{-0.1364 - 0.0152}{200} = -6.06 * 10^{-4} \text{ mm/mm}$$

STRESS

$$\sigma_1 = E\varepsilon_1 = -1.52 * 10^{-4} * 210 = -0.032 \text{ GPa}$$

$$\sigma_2 = E\varepsilon_2 = -6.06 * 10^{-4} * 210 = -0.127 \text{ GPa}$$

FORCES

$$f_1 = \sigma_1 A_1 = -0.032 * 314.16 = -10 \text{ KN}$$

$$f_2 = \sigma_2 A_2 = -0.127 * 78.54 = -10 \text{ KN}$$

This is a very simple example, but elaborately summarizes the finite element method used in a linear static solution. All the calculations demonstrated here are done by the solver. If the user needs detailed information about finite element method, it can be found in the online documentation or in the referenced books.

16.3 Linear Static Analysis Setup Using HyperMesh

There are some basic steps that the user needs to follow to setup a static analysis for OptiStruct, we will describe these steps using HyperMesh to setup the example described above. This example assumes that the OptiStruct BulkData user profile is used.

Step 1 - Define Material

1. From the Model Browser, right click and select Create > Material.
2. Then the user should input the material Name and select the card image (MAT1) and click on Create:

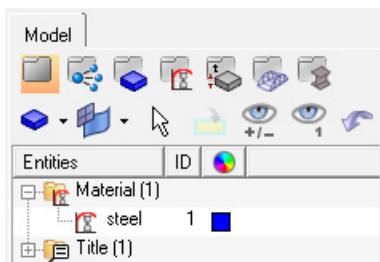
Name	Value
Solver Keyword	MAT1
Name	steel
ID	1
Color	█
Include File	[Master Model]
Card Image	MAT1
User Comments	Hide In Menu/Export
E	210000.0
G	
NU	0.3
RHO	7.85e-009

3. The material card now needs to be filled with the Young Modulus [E], Poisson [Nu], and optionally (dependent on the type of analysis carried out) Density [RHO] values and click return:

MAT 1	ID	[E]	[G]	[NU]	[RHO]	[A]	[TREF]	[GE]
	1	2.1e+05			0.300			
	[ST]	[SC]	[SS]					

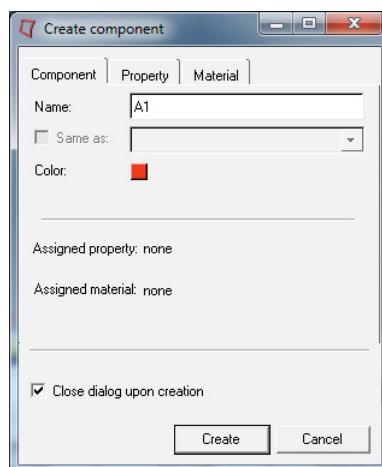
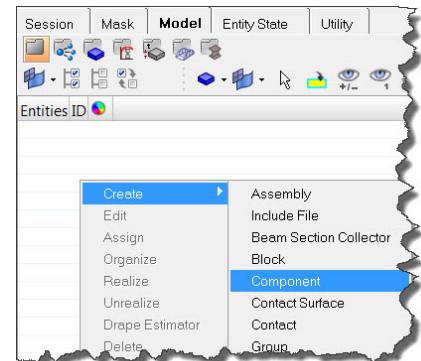
*If there are only 1D elements the Poisson value can be ignored.

The user should now see in the Model Browser the Material group with the material steel.

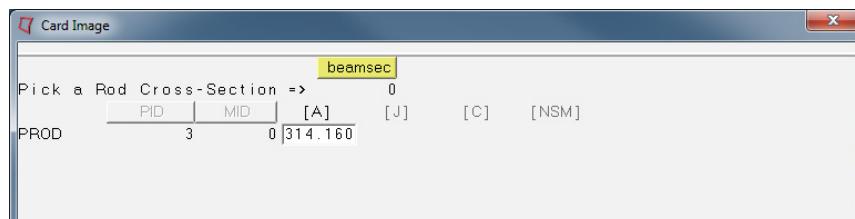


Step 2 – Create New Component- And Property Collectors

1. From the Model Browser, right click and select Create > Component.
2. Then the user should input the component Name as A1. Notice that there currently isn't an assigned property or material.
3. Click on the property tab and check the box for Assign property. For name enter A1 and for the card image select PROD. Select create property to enter the property card image.



4. Click on [A] to update the value of the area to 314.16 as shown below:



5. Click return to return to the create component dialog.

6. Click on the material tab and check the box for assign material. From the name drop down, select steel.

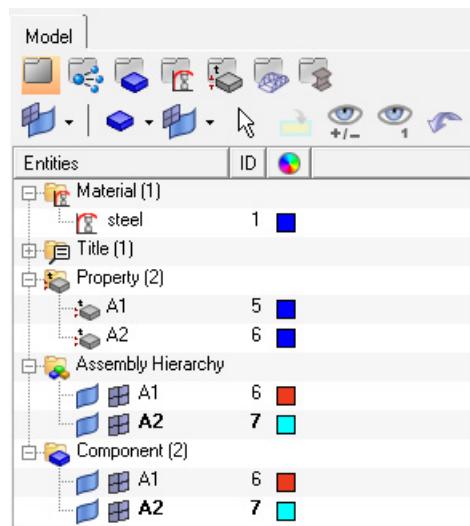
7. Click on the component tab and review the assigned property and material. Click create to create the property.

Name	Value
Name	A1
ID	3
Color	
Include File	[Master Model]
Property	A1 (1)
Material	steel (1)

8. Repeat the process to create a second component named A2 with a PROD property named A2 with an area of 78.54. Also assign the material steel. Also be sure to select a different color for the component.

Name	Value
Name	A2
ID	3
Color	
Include File	[Master Model]
Property	A2 (1)
Material	steel (1)

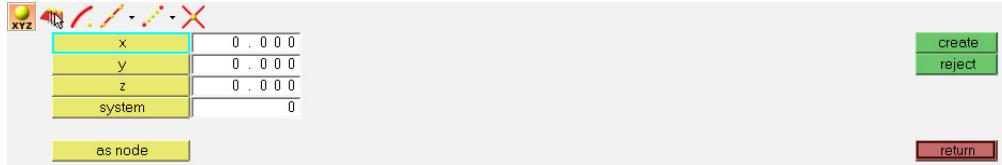
After creating these 2 components, you should see 2 new groups in the Model Browser called Component and Assembly Hierarchy, each with 2 instances called A1 and A2:



Step 3 – Create The Finite Element Mesh With The Appropriate Properties Associated To The Elements

This step is very model dependent. It can start with a CAD geometry that is imported or from other FEA Model, or can be a combination of existent models and imported geometries. In our case, this is a very simple model that we can model by creating the nodes and the respective elements.

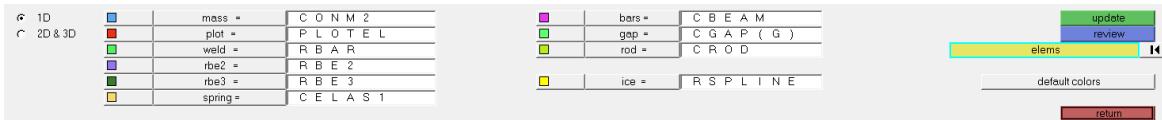
1. To create the nodes, use the shortcut key F8 that will open the Create Nodes panel:



2. Create three nodes at the following coordinates: (0, 0, 0), (100, 0, 0) and (300, 0, 0).



3. In our case it is necessary to create CROD elements. The user should use the Element Types panel (Mesh > Assign > Element Type) to set the rod element to CROD as below.



4. The Create rod panel can be accessed by the Mesh > Create > 1D Elements > Rods pull down menu.

5. Within the Create Rods panel, create a rod element between node 1 and node 2 in the A1 component with the A1 property (make the A1 component the Current Component using the Model Browser). Be sure to select the property before selecting the nodes to ensure that the appropriate property is assigned.

6. Repeat substep 5, making the component A2 the Current Component. Then create an element between node 2 and node 3 with the property A2.

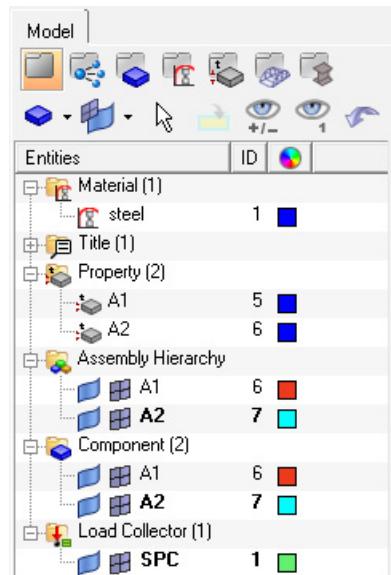


Step 4 – Define The Constraint Load Collector And Apply The Model Constraint

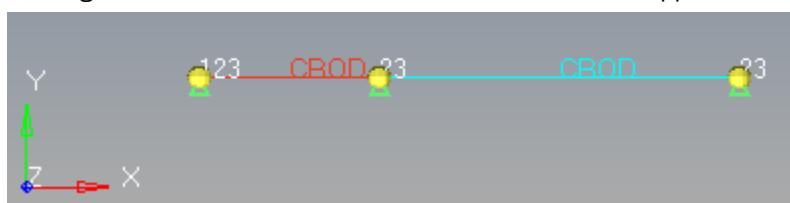
1. Right click in the Model Browser and select Create > LoadCollector to access the Create LoadCollector dialog box.
2. For Name, enter SPC, leave the Card image set to none and select Create.

Name	Value
Name	SPC
ID	1
Color	■
Include File	[Master Model]
Card Image	<None>

After creating this load collector, the user should see a new group called LoadCollector with an instance called SPC in the Model Browser.



3. Select BCs > Create > Constraints to create the SPC constraints. These constraints will fix the DOF that are not allowed to move.
4. In OptiStruct the rod element is a spatial element and has 3 DOFs (Ux, Uy and Uz). To reproduce the simple configuration we had on the first section, we need to remove all Uy and Uz DOFs and the Ux at node 1 as we did on the first section. Within the Constraints panel, select node 1 and check only the boxes for dof1, dof2, and dof3:
5. Select Create.
6. Repeat this process only selecting dof2 and dof3 for nodes 2 and 3. The model will appear as below:

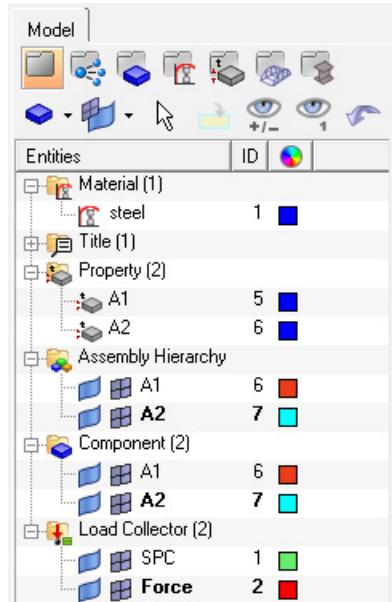


Now there are only 2 DOFs in this model Ux_2 and Ux_3 .

Step 5 – Define The Force Load Collector And Apply The Loads

1. Create a load collector named Force with no card image.

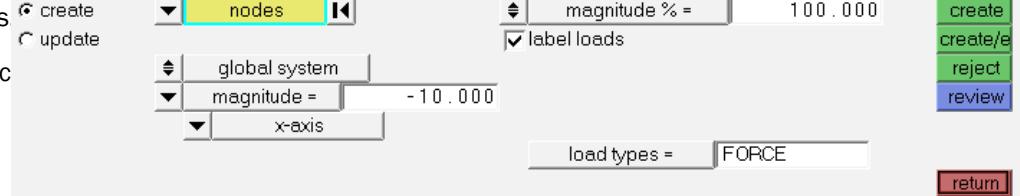
After creating this load collector, the user should see a new instance called Force on the LoadCollector group in the Model Browser:



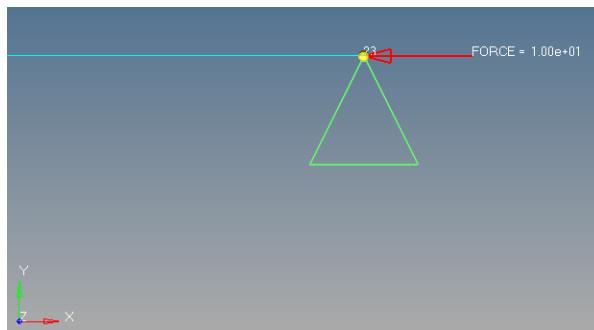
2. Now the user should create the force in the $-V$ direction at node 2 using the Forces panel. This panel is accessed from BCs >



3. For nodes, select



4. Select Create to create the force.



Step 6 – Define The Load Step

1. To create the load step, select Setup > Create > LoadSteps.

This will access the LoadSteps panel where the user should select the SPC and the load for a static load case and click on create

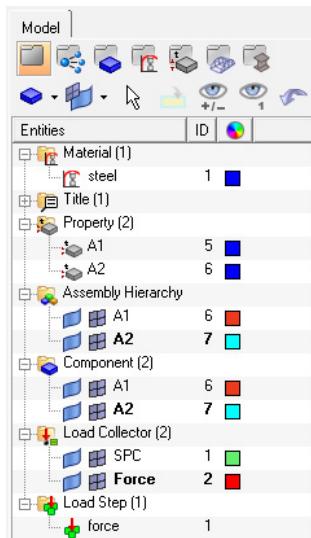
to create the load step.



2. For name enter force. Check the box for SPC and then click on the = and select the load collector SPC. Check the box for LOAD and then click on the = and select the load collector Force.

3. Verify that the type is set to linear static and select create.

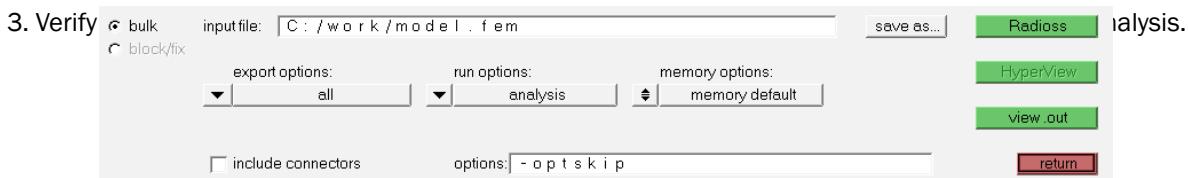
This will add a new group to the model browser tree called Loadstep with an instance called force:



Step 7 – Run The Analysis

1. Enter the Analysis page and then select the OptiStruct panel.

2. Click on “save as...” for input file and select a location and filename for the OptiStruct file using the extension *.fem.

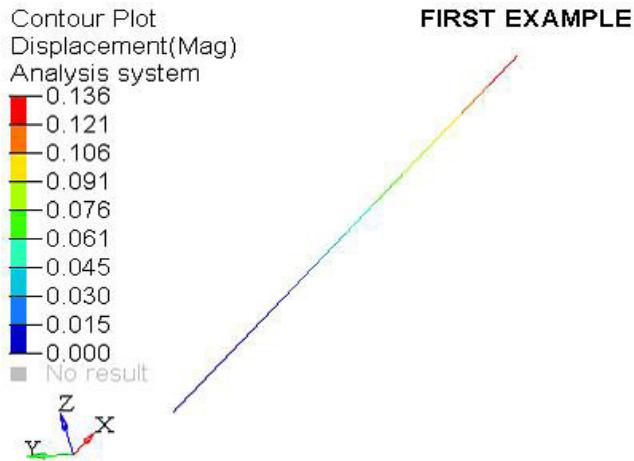


Step 8 – Post-Process The Results

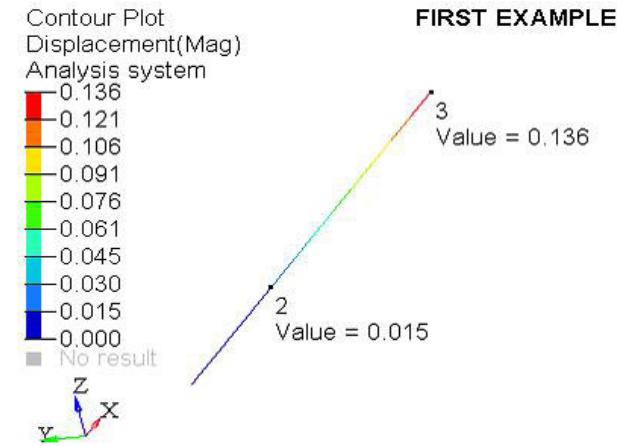
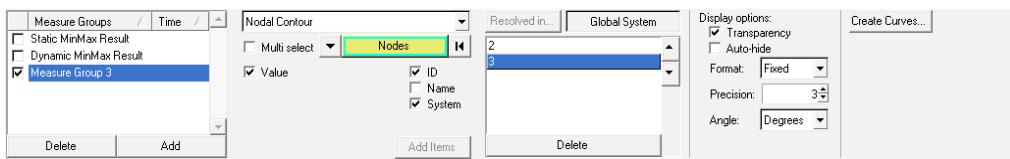
1. After the run is complete, the easiest way to access the results is by using the HyperView green button that is on the right hand side below the OptiStruct button used to call the solver.

This will add another page to the session with HyperView loaded as the client the resulting .h3d file loaded into HyperView.

2. In HyperView on the toolbar, click on the Contour icon, , select Displacement and Mag as the Result type and click Apply. This will generate the contour shown below:



3. The Measure panel, , can be used to compare this results with the one solved in the first section. Just click on "Add" and change the measure type to Nodal contour and select the nodes 2 and 3:



As we can see in the measures notes, the values match with the results evaluated earlier.

Many complex problems which can be approximated to a static system can be simplified and solved as a static analysis. Static analyses can also be used as a first check for nonlinear behavior, indicating a more involved simulation is required. The density of the materials is not needed, unless effects of mass of the components are desired. For a linear static analysis, when defining the material, one needs to specify either the Young's Modulus or the Bulk Modulus and the Poisson's Ratio. Care must be taken while evaluating the results from a static analysis to ensure that the linear assumptions are still valid. The following questions should be answered:

- Are the deflections and rotations small relative to the size of the structure being evaluated?
- Are there stress levels above the material yield?
- Is there any pass through of components with each other or with a boundary which would change the load paths through the system?

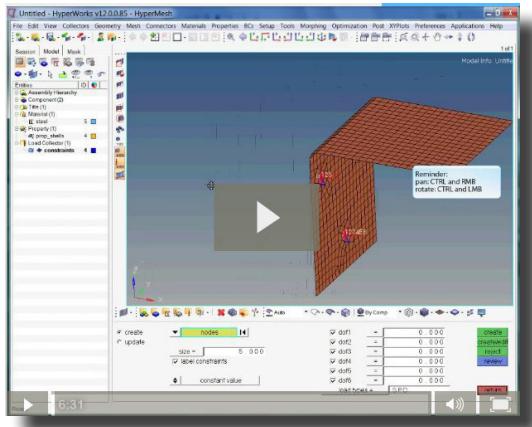
16.4 Linear Static Analysis Tutorials And Videos

Recommended Tutorials:

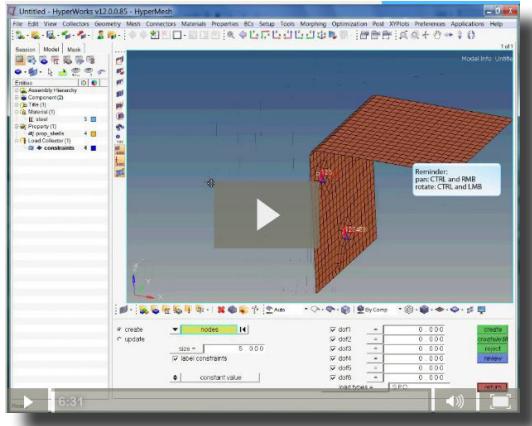
OS-1000: Linear Static Analysis of a Plate with a Hole (part of the HyperWorks Help documentation)

Recommended Videos

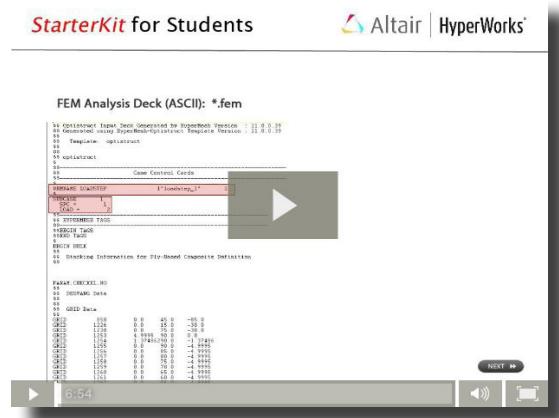
Loads (<http://altair-2.wistia.com/medias/tq0gr7zafo>)



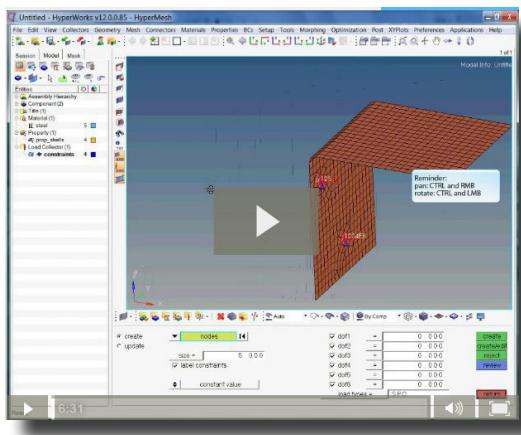
LoadStep/Subcase (<http://altair-2.wistia.com/medias/52c93wwc5x>)



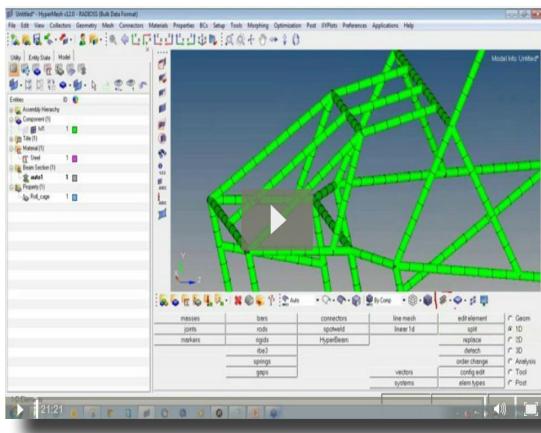
Input File (<http://altair-2.wistia.com/medias/3y89bitzjn>)



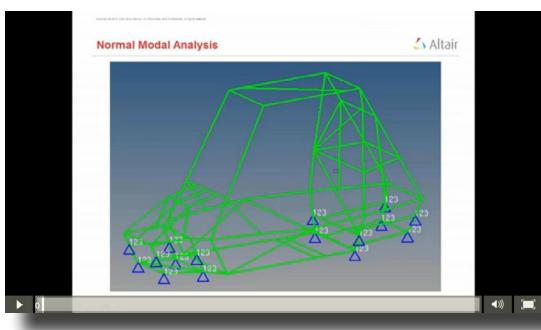
Log file, Error messages (<http://altair-2.wistia.com/medias/thkt5mhd79>)



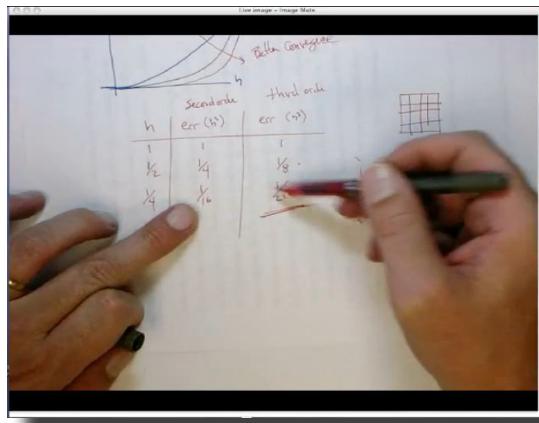
Torsional analysis of Roll Cage structure in RADIOSS/OptiStruct (<http://altair-2.wistia.com/medias/ek5p1mycxe>)



Normal modal analysis of Roll Cage structure in RADIOSS/OptiStruct (<http://altair-2.wistia.com/medias/lr7un6kib7>)



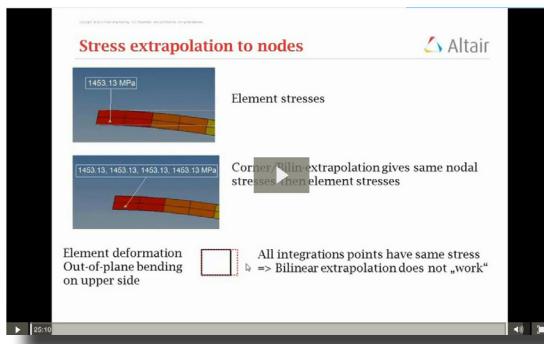
Some basic theory on the convergence of finite elements and how to determine the convergence rate theoretically and experimentally (published on Youtube by Prof. J. Chessa, Texas <http://youtu.be/YRJ0c8u08iU>)



Webinars

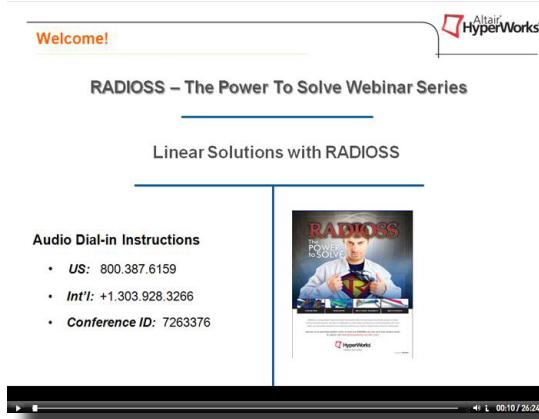
(note that the capabilities described in the webinars are now all part of OptiStruct)

HW BestPractice Webinars: Stress Calculation and Output in RADIOSS/OptiStruct



To view the other Webinars you need to login into the Client Center using your campus mail address

Linear Solutions with RADIOSS/OptiStruct (http://www.altairhyperworks.com/html/en-US/r1/Linear_Webinar1.aspx)



HW BestPractice Webinars: Output/Outfile Management for RADIOSS/OptiStruct

HW BestPractice Webinars: Using RBEs and MPCs in RADIOSS/OptiStruct

HW BestPractice Webinars: How to get a good quality FE model with RADIOSS/OptiStruct

16.5 Example: Analysis Of A Rotating Disc

The following illustrates the setup for the analysis of radial and tangential stresses of a “rotating” disc (linear static analysis).

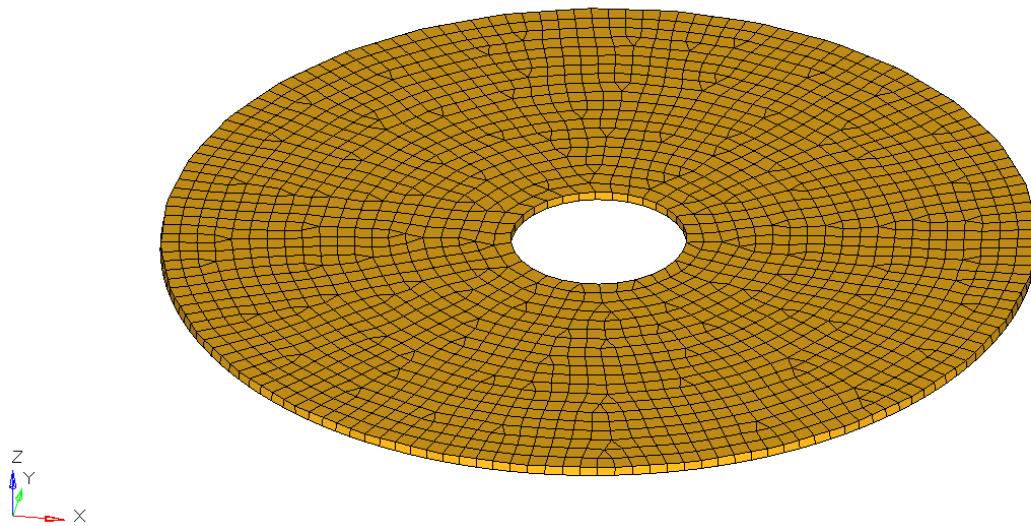
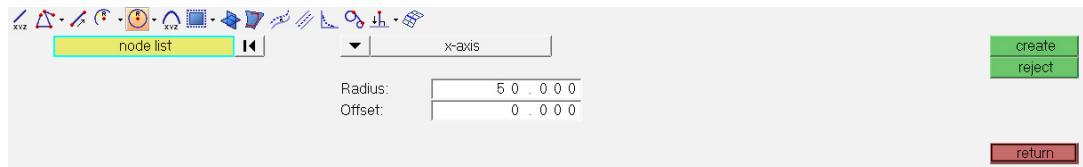


Figure: 3D FEM model of a “rotating” disc. $R_i=50$ mm, $R_o=250$ mm, Thickness $T= 5$ mm, material properties: $E= 210.000$ N/mm 2 , $\rho=7.9e-9$ t/mm 3 , average element size 10 mm

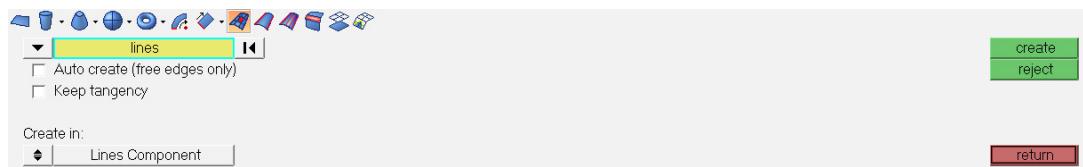
Geometry

The disc may be modeled by creating for instance:

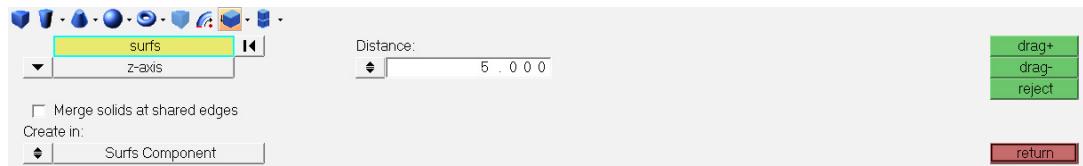
- 2 circles ($R_i = 50$ mm; $R_o=250$ mm)

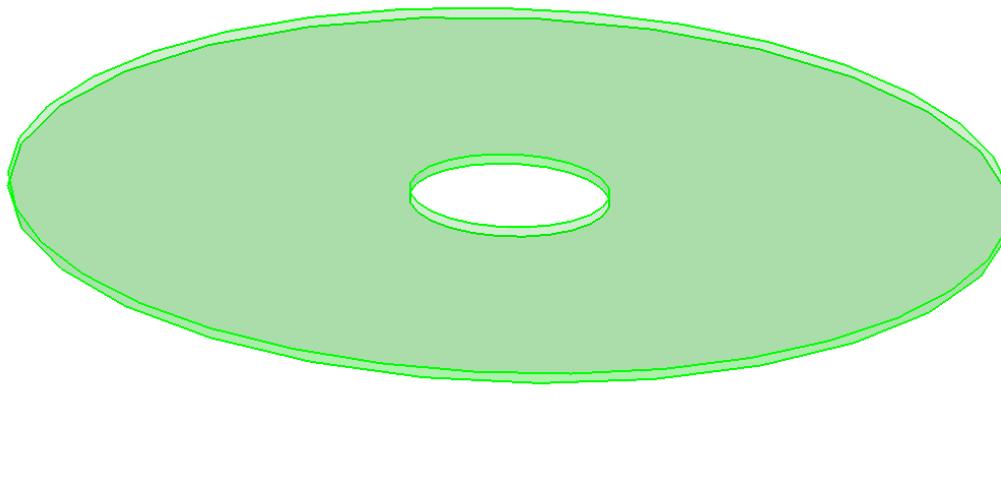


- Create a filler surface between the two circles



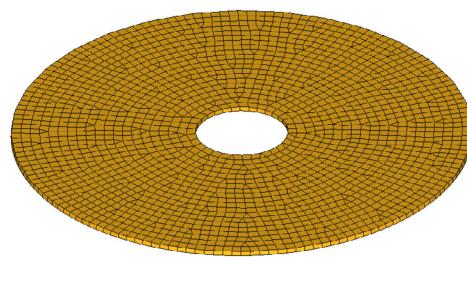
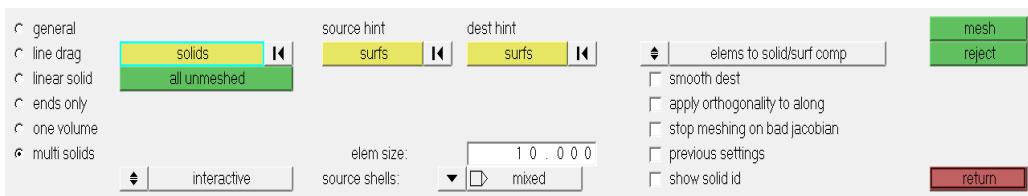
- Then drag the surface by 5 mm which results in a closed 3D geometry





Meshing

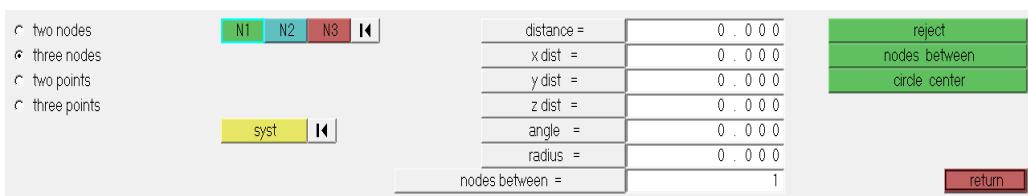
For the volume meshing the **Solid Map Mesh** panel may be used.



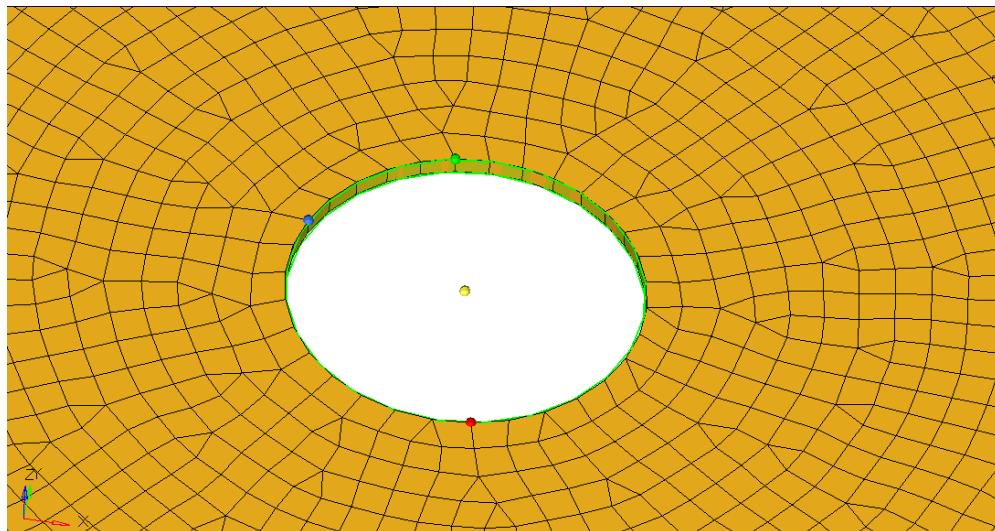
Define center of disc and cylindrical coordinate system

A cylindrical coordinate system needs to be assigned to the model in order to calculate the radial and tangential stresses (S_{xx} and S_{yy}).

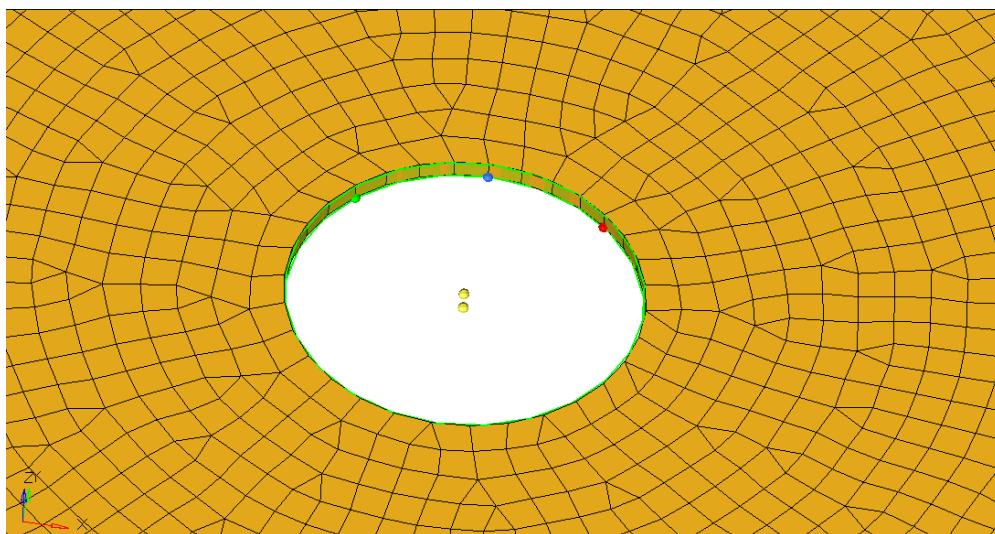
The center of the disc may be determined by making use of the Distance panel (note: The three nodes subpanel is active)



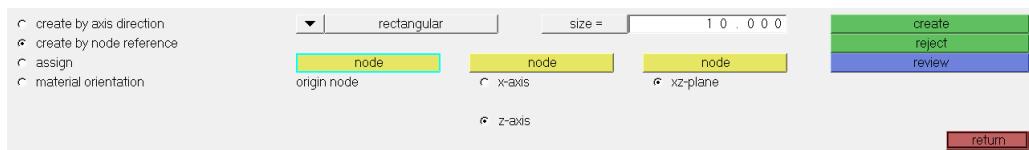
Just select three nodes on the inner hole and select circle center.

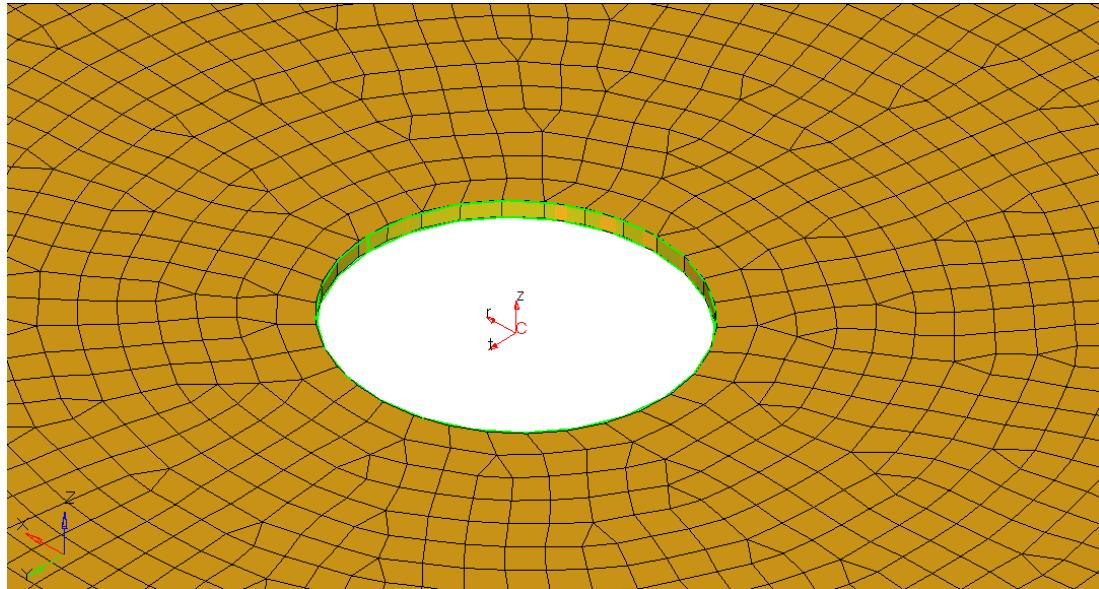
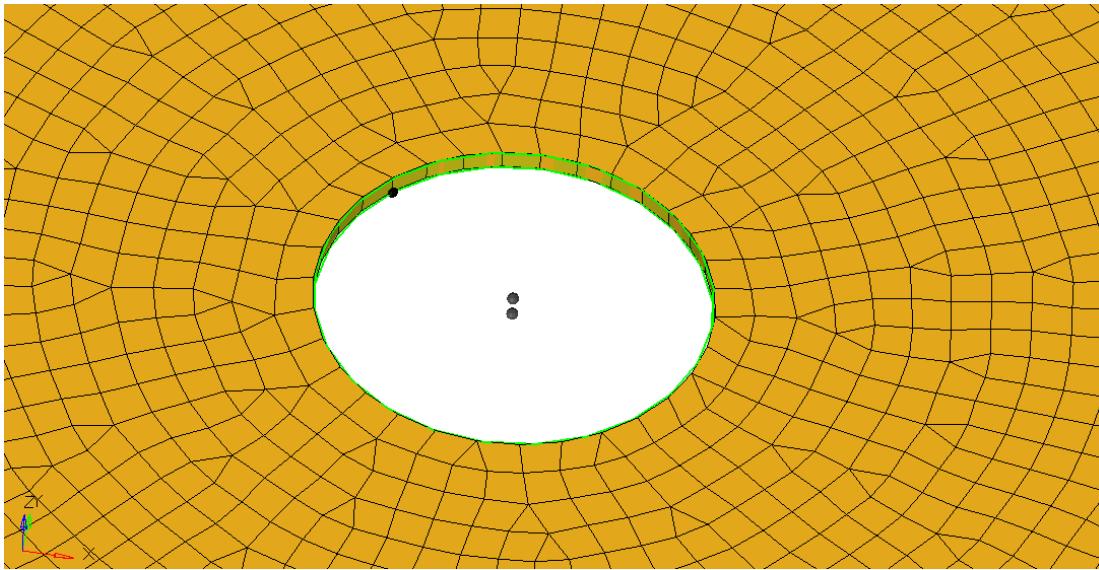


Repeat this last step for the lower edge of the hole



The two temporary nodes will be used to define the z-orientation of the cylindrical coordinate system. This is done by selecting Geometry > Create > Systems > Node Reference.





This newly created cylindrical coordinate system must be assigned to all nodes of the model. To do this, select all nodes, then select graphically the local coordinate system (you need to click on it) and select: set displacement.

<input type="radio"/> create by axis direction	set:	<input type="button" value="nodes"/>	size =	1 0 . 0 0 0	<input type="button" value="set reference"/>
<input type="radio"/> create by node reference					<input type="button" value="set displacement"/>
<input checked="" type="radio"/> assign					<input type="button" value="review"/>
<input type="radio"/> material orientation	to:	<input type="button" value="system"/>			<input type="button" value="preview unused"/>
					<input type="button" value="delete unused"/>
					<input type="button" value="return"/>

Boundary Condition And Loads

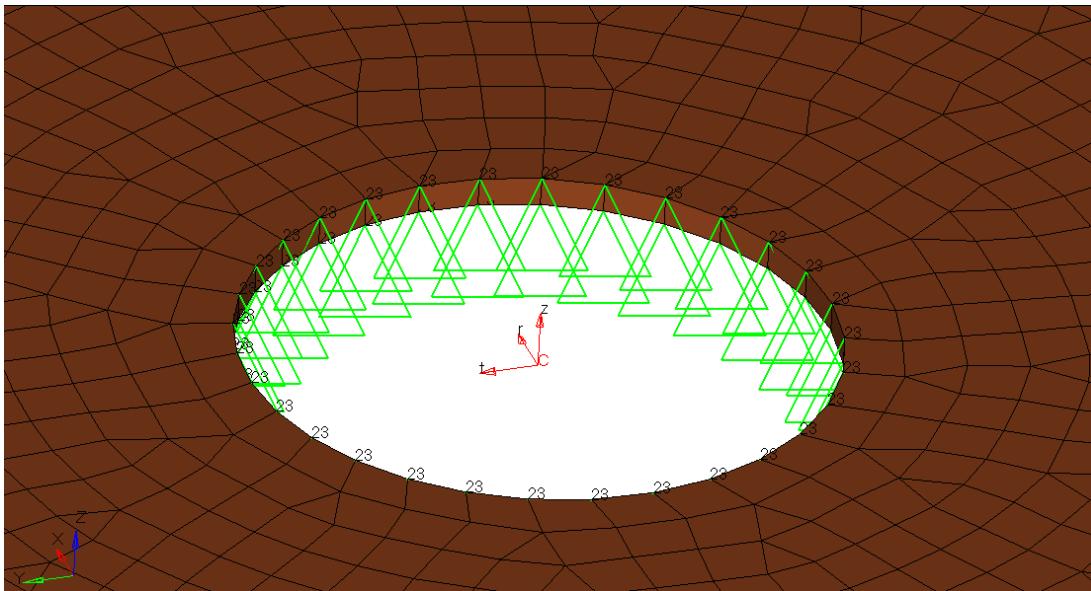
The disc will be constrained at its inner hole (inner radius) by fixing/deleting the translational displacements in the tangential and axial direction. The radial displacements are not constrained (free boundary).

Dof 1 (radial): free, no constraints

Dof 2 (tangential) = 0

Dof 3 (axial) = 0

Remember: The rotational dofs are not available with solid elements



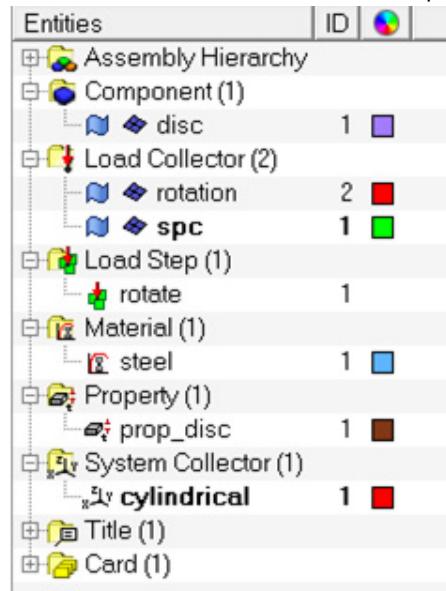
The rotational forces due to a constant angular velocity of the disc are created by means of a specific card image named RFORCE (rotational forces act in the positive radial direction). Note: the modeled disc is not really rotating as this is a static analysis.

Thus, create another load collector (in this example it is named: rotation) with the card image RFORCE.

R FORCE	SID	G	CID	[A]	R1	R2	R3
	2	1	0	39.789	0.000	0.000	1.000
	RACC			0.000			

RFORCE card image: SID refers to the ID of the load collector, G defines the node through which the rotation vector acts, CID typically refers to the global Cartesian coordinate system, A is the rotational velocity in revolutions per unit time (here 39.789 revolutions per second = 250 rad/second); R1-R3 define the rotation axis (here rotation axis is z-direction).

Eventually, create a static load step based on the two load collectors named "spc" and "rotation".



Material And Property

Create a material collector using the standard values for steel:

MAT 1	ID	[E]	[G]	[NU]	[RHO]
	1	2.1e+05			7.9e-09
	[ST]	[SC]	[SS]		

While creating the property collector, keep in mind that the elements are solid elements

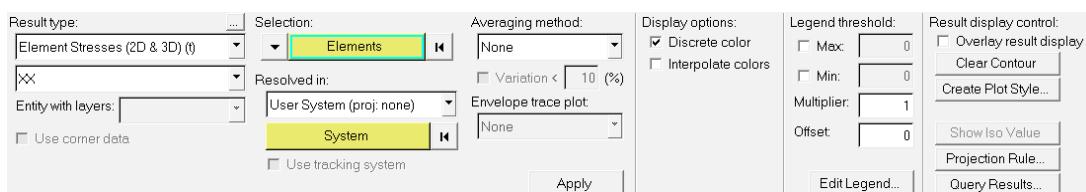
PSOLID	PID	MID	
	1	1	

Then, assign the property (which internally references the material) to either the elements or their corresponding collector. It is up to you.

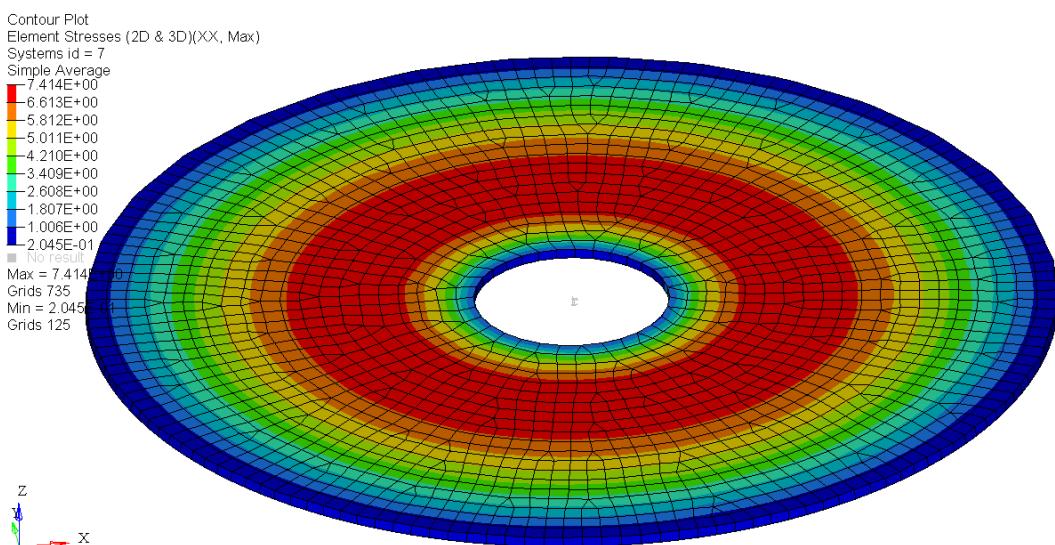
That's it - you are ready to start the analysis.

Post-Processing

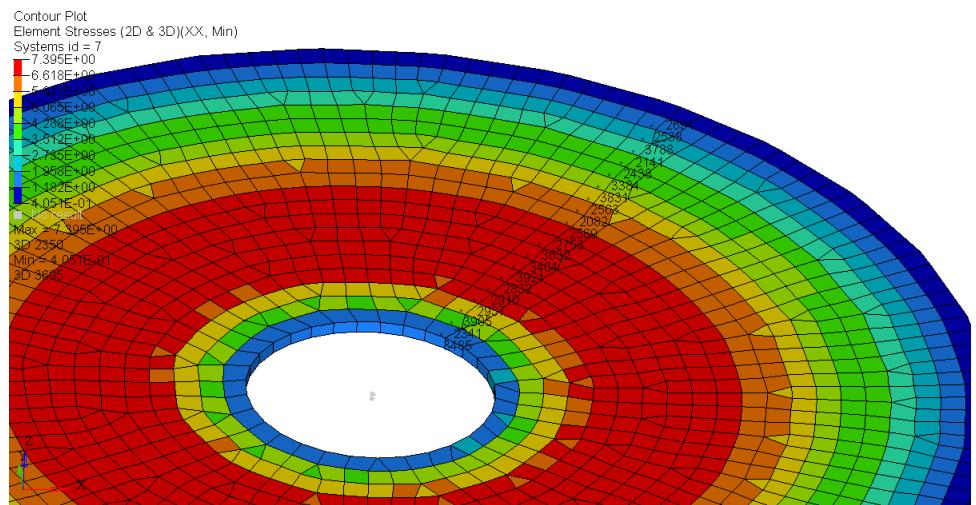
Make sure that when creating the stress contour plots that the local cylindrical coordinate system is selected. To do this, select under "Resolved in" the System ID of the cylindrical coordinate system.



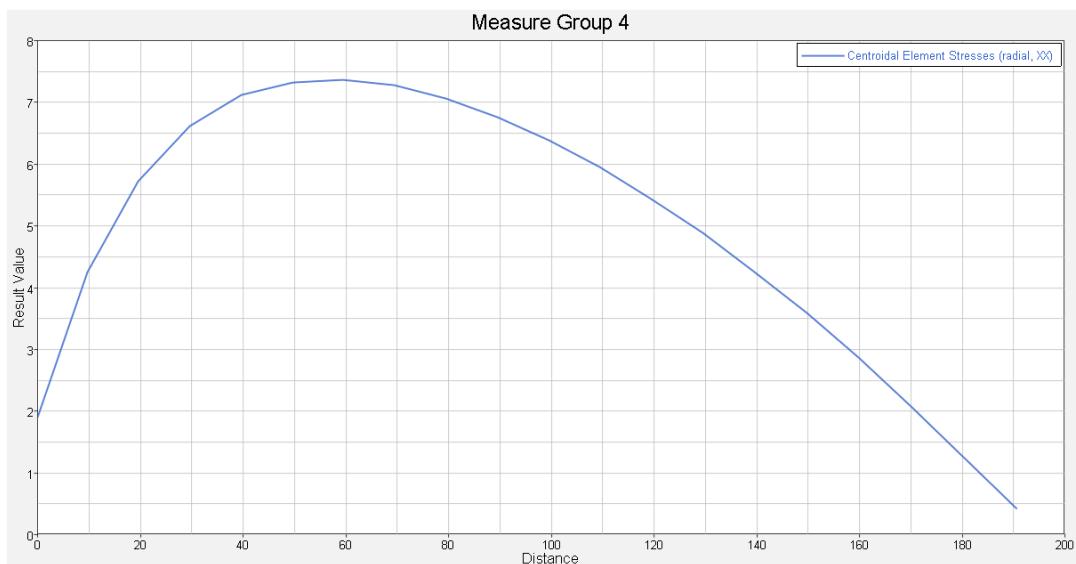
Otherwise the Sxx will be normal stresses in Cartesian coordinates.



Radial averaged element stress.



In the image above the element path used for plotting the elemental stresses (at the centroid) versus distance is shown.



Radial stresses (at element centroid). Distance is measured from element centroid to element centroid, with an average element size of 10 mm.

16.6 Student Racing Car Project: Linear Static Analysis

As the UserProfile (hence the FEM solver) is chosen to be OptiStruct, we start the analysis from within the panel OptiStruct (Optimization > OptiStruct):



Take care and don't forget to add the file extension *.fem. The "run options" is set to "analysis".

Before visualizing displacements and stresses, check the *.out file located in the directory where the analysis file was saved in.

```

File Edit
Label x-force y-force z-force x-moment y-moment z-moment
-----
Sum-App. 0.000E+00 0.000E+00 0.000E+00 2.109E+06 -3.691E+04 0.000E+00
Sum-SPCF -1.450E-07 7.283E-07 -1.046E-09 -2.109E+06 3.691E+04 -5.387E-04

Notes: 1. All applied and SPC forces are transferred to
       the origin of the basic coordinate system,
       so that the applied and SPC loads can match.
       2. If spring elements and/or MPCs exist in the model,
          total applied loads may not match total SPC loads.
(Running in-core solution)

Volume = 5.33164E+06 Mass = 4.21200E-02

Subcase Compliance
 2 1.371717E+03
 3 9.762321E+02
 4 1.170128E+04

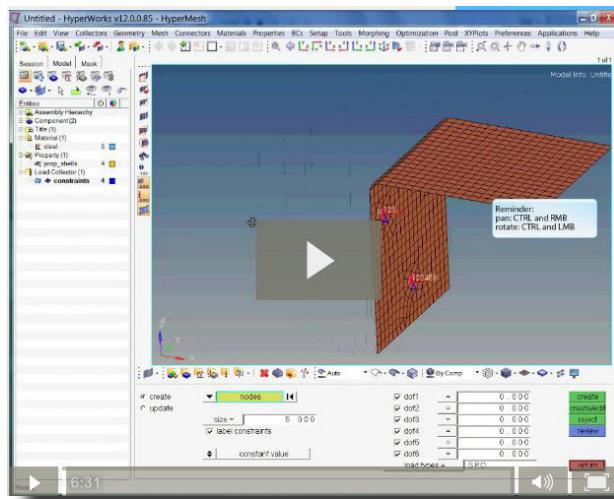
***** ANALYSIS COMPLETED. *****

```

In the outfile a model summary, estimated memory, likely warning and errors etc. are documented.

Hint: The most typical error messages and debugging steps related to a linear static analysis are discussed in the video:

Debugging (<http://altair-2.wistia.com/medias/thkt5mhd79>)



17 Modal Analysis

This chapter is based on the HyperWorks Help Documentation

17.1 Introduction

Normal Modes Analysis, also called eigenvalue analysis or eigenvalue extraction, is a technique used to calculate the vibration shapes and associated frequencies that a structure will exhibit. It is important to know these frequencies because if cyclic loads are applied at these frequencies, the structure can go into a resonance condition that will lead to catastrophic failure. It is also important to know the shapes in order to make sure that loads are not applied at points that will cause the resonance condition.

Normal modes analysis is also required for modal frequency response and modal transient analysis. In these analyses the problem is transformed from the direct mesh coordinates, where the number of degrees of freedom can be in the millions, to the modal coordinates where the number of degrees of freedom is just the number of modes used. Typically, the upper bound frequency in this case is 1.5 times the highest loading frequency or response frequency of interest.

The Governing Equations - Normal Modes Analysis

The equilibrium equation for a structure performing free vibration appears as the eigenvalue problem:

$$[K - \lambda M]x = 0$$

where K is the stiffness matrix of the structure and M is the mass matrix. Damping is neglected.

The solution of the eigenvalue problem yields n eigenvalues λ_i , where n is the number of degrees of freedom. The vector { Φ_i } is the eigenvector corresponding to the eigenvalue λ_i .

The eigenvalue problem is solved using **Lanczos**, **AMSES**, or **AMLS** (see further down for details)

The natural frequency f_i follows directly from the eigenvalue λ_i .

$$f_i = \frac{\sqrt{\lambda_i}}{2\pi}$$

In OptiStruct, normal modes analysis can be performed using one of two algorithms: Lanczos or the automated multi-level sub-structuring eigenvalue solution (AMSES). The eigenvalue extraction data for Lanczos is specified on the EIGRL data and for the automated multi-level sub-structuring eigenvalue solution method, the EIGRA data is used (EIGRL and EIGRA are explained further below).

In addition, OptiStruct has an interface to the AMLS software developed at the University of Texas. AMLS uses the automated multi-level sub-structuring method for eigenvalue extraction. The use of AMLS is triggered by using the PARAM AMLS set to YES input data in conjunction with the EIGRL card (only).

The Lanczos Method

The Lanczos method has the advantage that the eigenvalues and associated mode shapes are calculated exactly. This method is efficient for calculations in which the number of modes is small and the full shape of each mode is required. The disadvantage of the Lanczos method is that it is slow for large problems with millions of degrees of freedom for which hundreds of modes are required. The run times for these types of problems can easily stretch into days. In these cases, the AMSES or AMLS method must be used.

The Automated Multi-Level Sub-Structuring Eigenvalue Solution Method (AMSES)

The AMSES method has the advantage that only a portion of the eigenvector need be calculated. Since only a portion of the eigenvector is calculated, the disk space and disk I/O is greatly reduced. This leads to much shorter run times. For typical NVH frequency response analysis there is only about 100 degrees of freedom of interest. In these cases, solutions of thousands of modes for meshes of millions of degrees of freedom can be solved in just a few hours. The disadvantage of the AMSES method is that the calculations are not exact. However, the modal frequencies are still accurate to a few digits. Also, for NVH analysis it is important that the mode shapes form modal space that covers all possible deformation patterns, but not so important that each individual mode shape is accurate.

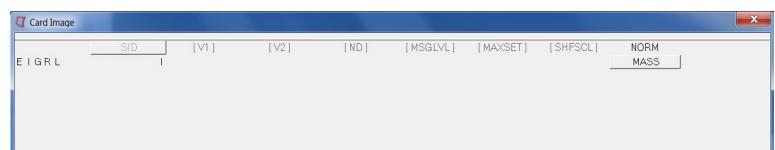
More information about AMSES Usage is included in the HyperWorks Help Documentation.

Input Specification

In order to run a normal modes analysis, an EIGRL or EIGRA bulk data entry needs to be given to define the number of modes to be extracted. EIGRL or EIGRA data needs to be referenced by a METHOD statement in a SUBCASE in the subcase information section.

EIGRL and EIGRA are so-called Card Images of a corresponding load collector

Name	Value
Name	Loadcollector1
ID	1
Color	Blue
Include File	[Master Model]
Card Image	<None>
	<None>
	ACSRCE
	CAALOAD
	CDSMETH
	CMSMETH
	DLOAD
	DTI
Optistruct	EIGC
	EIGRA
	EIGRL



More information about the EIGRL Card Image is also included in the tutorial “[Modal Analysis with Altair OptiStruct / HyperMesh](#)” at the end of this chapter).

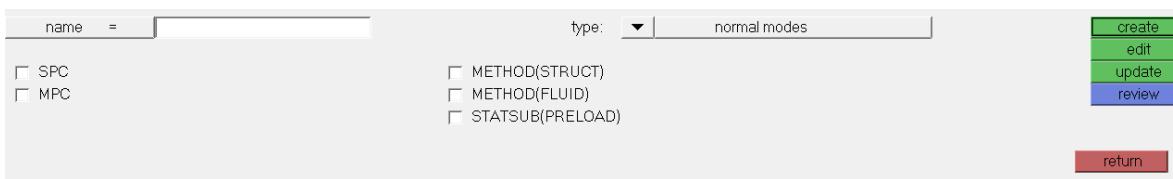
It is not necessary to define boundary conditions using an SPC statement (SPC = Single Point Constraint). If no boundary conditions are applied, a zero eigenvalue is computed for each rigid body degree of freedom of the model.

Subcase Definition (Loadstep)

A normal modes subcase may be explicitly identified by setting ANALYSIS=MODES, but it is also implicitly chosen for any subcase containing the METHOD data selector (when the ANALYSIS entry is not present).

In the loadstep/subace panel (Setup-->Create-->Loadsteps) activate the toggle “type”. Thsi opens up a window with a selection of available analysis types

linear static	freq. resp. (direct)	complex eigen (model)	geom non-linear (impl dynamic)
heat transfer (steady state)	freq. resp. (model)	multi-body dynamics	response spectrum
heat transfer (transient)	transient (direct)	fatigue	combination subcase delimiter
normal modes	transient (model)	geom non-linear (impl static)	nonlinear heat transfer
linear buckling	non-linear quasi-static	geom non-linear (expl dynamic)	generic



The following data selectors are recognized for an normal modes subcase definition.

1. METHOD – references an eigenvalue extraction bulk data definition (EIGRL or EIGRA). This reference is required.
2. SPC – references single point constraint bulk data entries (SPCADD or SPC and SPC1).
3. MPC – references multi-point constraint bulk data entries (MPCADD or MPC).

Bulk Data

Bulk data entries which have particular significance for normal modes analysis include:

1. EIGRL – specifies the modes to be calculated and solution parameters for the Lanczos eigenvalue extraction method.
2. EIGRA – specifies the modes to be calculated and solution parameters for the AMSES eigenvalue extraction method.
3. PARAM,AMLS,YES – specifies that the AMLS software will be used for eigenvalue extraction based on the modal parameters on the EIGRL or EIGRA data.
4. SPC, SPC1, SPCADD - specify the base where excitation is applied and other constraints.
5. MPC, MPCADD - specify multi-point constraints.

Output

Results of interest from eigenvalue extraction include maximum displacement, modal stresses, energies and multi-point constraint forces. These are requested via the I/O Options DISPLACEMENT, EKE, ESE, STRESS, GPSTRESS and MPCFORCE respectively.

Let's have a look at the model setup in the following tutorial / project.

17.2 Example: Modal Analysis

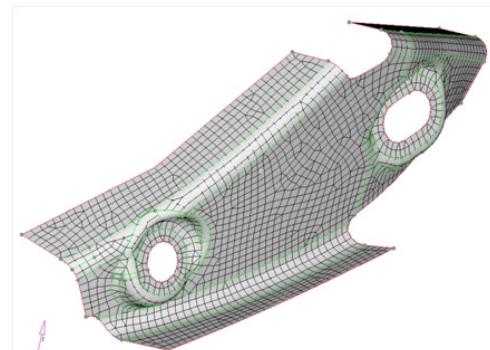
This example originally comes from Tony Gray, modified by Moritz Guenther, Jacob Tremmel, Rahul Ponginan and Prakash Pagadala

Some Hints

All components in the model must have material and properties assigned/defined. Make sure units are consistent and density is defined.

(Example - if model is in mm for Steel then: Youngs Modulus = 210.000 MPa, Density = 7.9e-9 t/mm³)

Modal analysis is typically a free or constrained model. A free analysis doesn't require constraints but will generate rigid body modes. You can avoid extracting these using variables on the EIGRL card.



Let's Get Started - Work Flow

- Start HyperMesh and set the user profile to OptiStruct
- Open the model: model_start.hm
- Mesh components, create/define material and property
- Define constraints (if desired) and modal extraction load collector
- Create load step / subcase
- Define any special output requests and/or control cards

Meshing

Mesh the model using "AutoMesh" panel with element size of approx. 2mm (mixed element types)

1. Access the "Automesh" panel (Menu Bar>Mesh>Create>2D AutoMesh)
2. Select surfs >> displayed
3. Element size = 2 mm; mesh type: mixed.
4. Meshing mode "interactive" (lower left corner)
5. Ensure that "elems to current comp" is active.
6. Click "mesh" to enter the meshing subpanel

(Notice that you are in the density subpanel of the automesh panel. Mesh density changes etc. could be made there if necessary)

7. Click "return" to accept the mesh as the final mesh

Material Definition

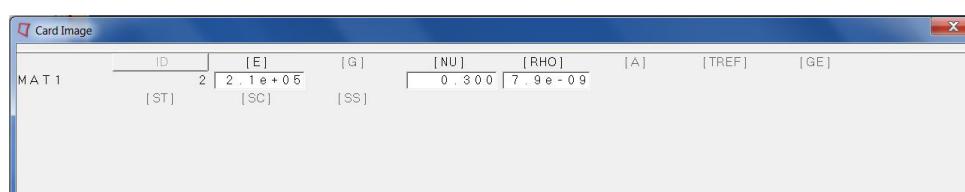
Create a material of type MAT1 (Card image), named "steel"

1. Right click in Model Browser>Create>Material

2. Enter name = steel
3. Click type = and select ISOTROPIC
4. Click card image = select MAT1
5. Activate the checkbox next to “Card edit material upon creation”
6. Click “create”



(Notice: If a material property in brackets does not have a value below it, it is off. To edit these material properties, click the property in brackets you wish to edit and an entry field will appear below it. Click the entry field and enter a value)



7. Accept the standard values prompted

E as 2.1e5 N/mm²

NU as 0.3

RHO as 7.9e-09 t/mm³

8. Click return

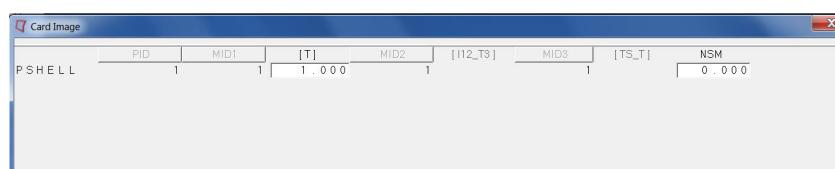
A new material, named “steel”, has been created. The material uses OptiStruct’s linear isotropic material model, MAT1. This material has a Young’s Modulus of 2.1e+05 MPa, a Poisson’s Ratio of 0.3, and a density of 7.9e-9 t/mm³.

At any time, the card image for this collector can be viewed & modified with right mouse click on the material of interest (or more general, the collector of interest). In the pop-up window select “Card Edit”

Property Definition

Create a property collector with Card image “PSHELL”, named “prop_model” with thickness 1mm.

Follow the process as described with respect to the material definition and enter the corresponding values. Make sure that the previously defined material is referenced.



Finally assign the property collector to all elements:

Right click on property collector “prop_model” in the Model Browser, then select “Assign” ...

Loads

Create a load collector with Card image EIGRL, named “freq”.

The working steps are similar to the previous ones. The Card Image EIGRL is displayed below with V1 = 1.0 and ND = 3.



V1&V2 define the range you wish to extract the modes over. ND defines the number of modes you wish to find. Various combinations of these cards can be used to control what is extracted. This combination extracts the first 3 modes above 1 Hz. Setting V1 to 1.0 is a “trick” to avoid the extraction of rigid body modes.

Note: MORM specifies the Method used for eigenvector normalization. If MASS, then eigenvectors are normalized to the unit value of the generalized mass (this is not a valid option for linear buckling analysis). If MAX, then eigenvectors are normalized to the unit value of the largest displacement in the analysis set. Default = MASS for normal modes analysis; Default = MAX for linear buckling analysis (MASS or MAX).

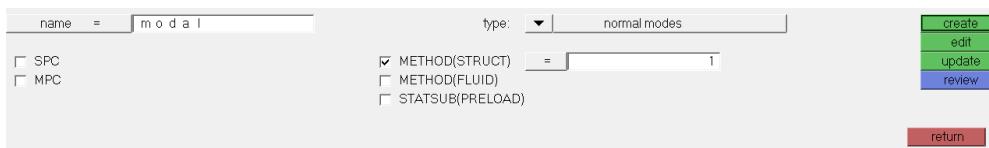
If you use the default (MASS) for normal mode analysis, then the deformed plot in HyperView may show very high “displacements”. This is because HyperView displays deformations (by default) using the scale option “Model units”. Thus simply use a scale factor < 1 to display the deformed structure. Alternatively, change “Scale: Model units” to “Model percent”.

Finally, click return.

Loadstep / Subcase

Create a subcase called “modal” which refers to the EIGRL card defined before. This is done using the METHOD card as shown below.

1. Click Setup > Create > LoadSteps
2. name =modal; type = normal modes
3. Activate METHOD (STRUCT) and select the load collector with the EIGRL card image.



OUTFILE	PROPERTY	SHAPE	delete
OUTPUT	RESPRINT	SHRES	enable
P2G	RESTART	SUBTITLE	
PARAM	RESULTS	SWLDPRM	
PFGRID	SCREEN	SYSSETTING	
PFMODE	SENSITIVITY	THICKNESS	next prev

Analysis

In the Control Cards (located on page Analysis >Control Cards>) set the SCREEN output to OUT

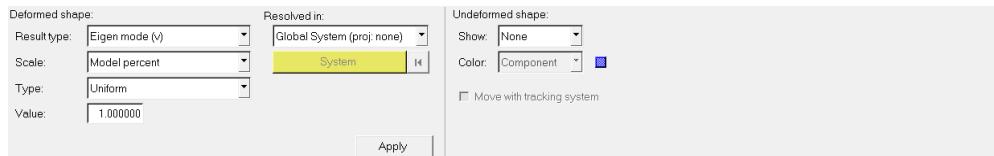
Save the model then run the analysis from the OptiStruct panel (Analysis > Optistruct).

You should see the following results. Note the first frequency is 429 Hz.

Subcase	Mode	Frequency	Eigenvalue	Stiffness	Mass
1	1	4.296100E+02	7.286324E+06	7.286324E+06	1.000000E+00
1	2	8.960945E+02	3.170059E+07	3.170059E+07	1.000000E+00
1	3	1.429868E+03	8.071453E+07	8.071453E+07	1.000000E+00

Start HyperView, load the corresponding *.h3d file and open up the “deformed panel”.

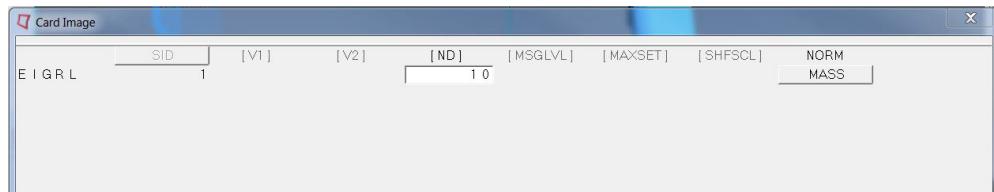
Change “Scale: Model units” “Model percent”, then click “Apply”.



You will be able to animate the 3 mode shapes using the modal button.

Annotation / Remark

Edit the EIGRL card by removing V1 and set ND to 10 (= the first 10 modes).

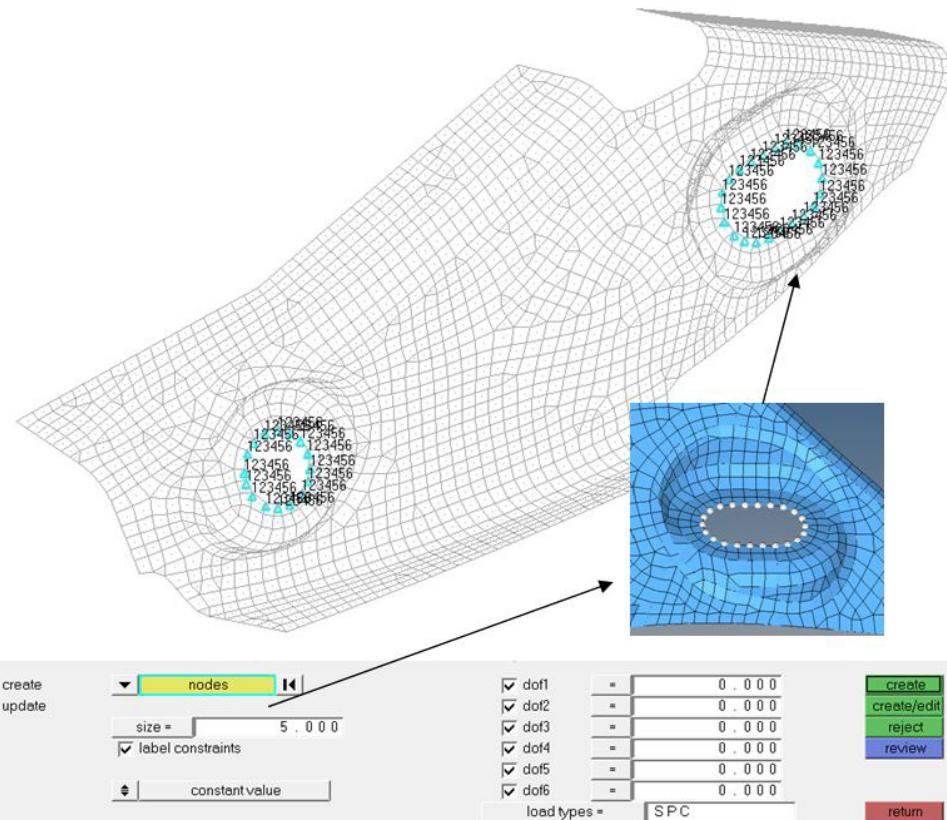


Re-run the model. Note the first 6 modes are now very small – these are the 3 translational and 3 rotational degrees of freedom.

Subcase	Mode	Frequency	Eigenvalue	Stiffness	Mass
1	1	6.797740E-03	1.824269E-03	1.824269E-03	1.000000E+00
1	2	7.368257E-03	2.143331E-03	2.143331E-03	1.000000E+00
1	3	7.779286E-03	2.389127E-03	2.389127E-03	1.000000E+00
1	4	7.857455E-03	2.437382E-03	2.437382E-03	1.000000E+00
1	5	8.119594E-03	2.602725E-03	2.602725E-03	1.000000E+00
1	6	8.581614E-03	2.907352E-03	2.907352E-03	1.000000E+00
1	7	4.296100E+02	7.286324E+06	7.286324E+06	1.000000E+00
1	8	8.960945E+02	3.170059E+07	3.170059E+07	1.000000E+00
1	9	1.429868E+03	8.071453E+07	8.071453E+07	1.000000E+00
1	10	1.776882E+03	1.246455E+08	1.246455E+08	1.000000E+00

17.3 Example: Modal Analysis With Constraint Model

1. Create another load collector (using no card image) called “constraints”
2. From Model Browser expand LoadCollectors, right-click on constraints and click “Make Current” to set constraints as the current load collector.
3. Fully constrain (all 6 dofs) the nodes around each hole as illustrated in the image below.



- Click BCs > Create > Constraints to open the Constraints panel.
- Select the nodes around the two holes
- Constrain dof1, dof2, dof3, dof4, dof5, and dof6 and set all of them to a value of 0.0.

Note:

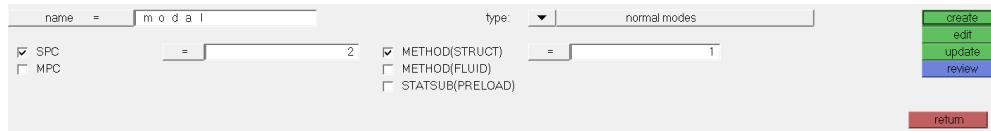
Dofs with a check will be constrained, while dofs without a check will be free.

Dofs 1, 2, and 3 are x, y, and z translation degrees of freedom.

Dofs 4, 5, and 6 are x, y, and z rotational degrees of freedom.

- Click create
- Click return to go to the main menu
- Update the existing loadstep/subcase to include this SPC (= Single Point Constraint)

- Click Setup > Create > LoadSteps to open the LoadSteps panel.
- Select SPC and then click on the field next to SPC and select the constraints load collector created above,



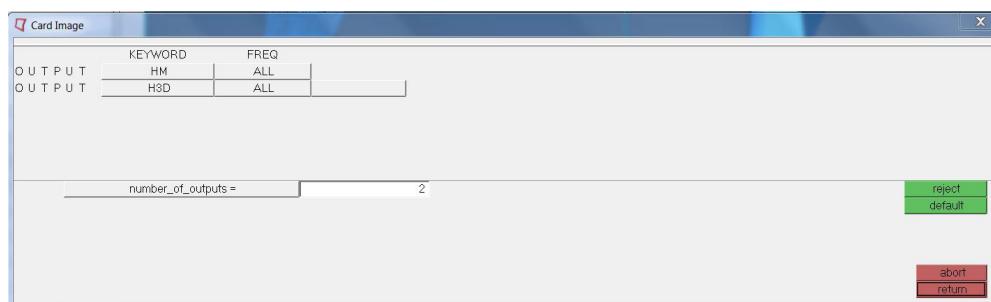
Finally, re-run the analysis. Note that the rigid body modes are gone.

Subcase	Mode	Frequency	Eigenvalue	Stiffness	Mass
1	1	1.349106E+03	7.185419E+07	7.185419E+07	1.000000E+00
1	2	1.847394E+03	1.347345E+08	1.347345E+08	1.000000E+00
1	3	1.859691E+03	1.365342E+08	1.365342E+08	1.000000E+00
1	4	2.034115E+03	1.633468E+08	1.633468E+08	1.000000E+00
1	5	2.111028E+03	1.759332E+08	1.759332E+08	1.000000E+00
1	6	2.611230E+03	2.691846E+08	2.691846E+08	1.000000E+00
1	7	2.806940E+03	3.110470E+08	3.110470E+08	1.000000E+00
1	8	3.123768E+03	3.852274E+08	3.852274E+08	1.000000E+00
1	9	3.228458E+03	4.114812E+08	4.114812E+08	1.000000E+00
1	10	3.552249E+03	4.981574E+08	4.981574E+08	1.000000E+00

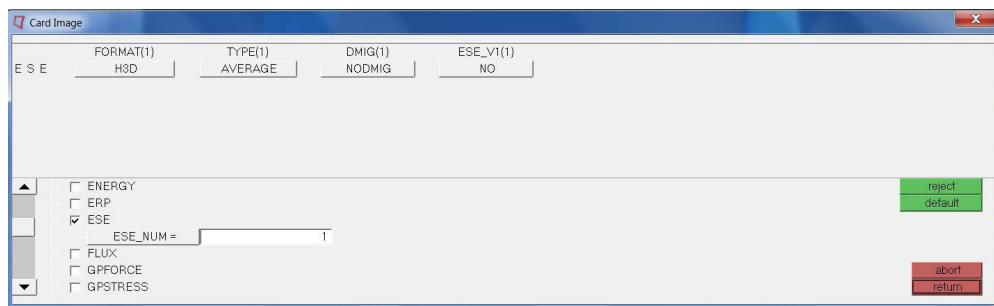
17.4 Tips & Tricks Regarding Post-Processing

Result Information Which Needs To Be Requested Prior To The Analysis

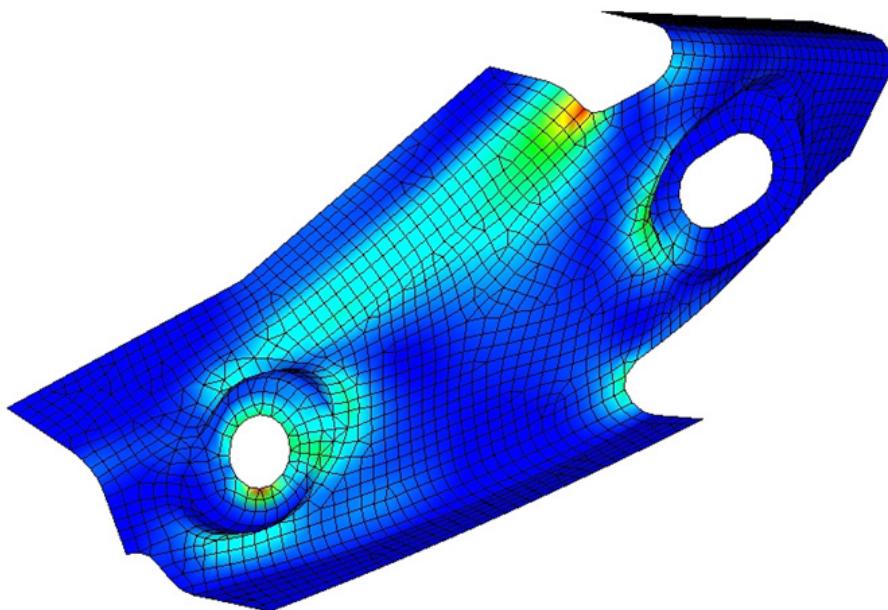
Sometimes you want more data from your analysis or your results in different formats. Note that by default OptiStruct creates an HTML file summarizing the analysis and writes a HyperMesh results file. To get results in other formats e.g. H3D use the output control card (Analysis > Control Cards >....).



Another output that is often useful in modal analysis is element strain energies. You can request these using the ESE function of the GLOBAL_OUTPUT_REQUEST control card (Analysis > Control Cards >).

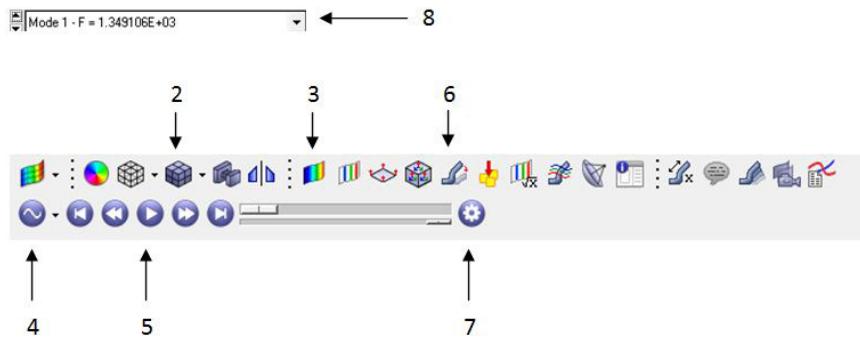


The strain energy results allow you to identify areas critical to given modes

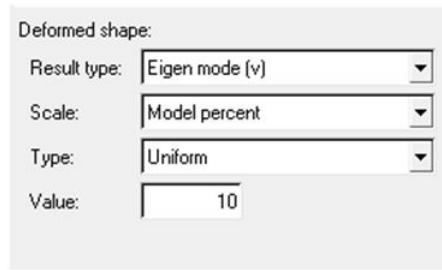


Post-Processing With HyperView

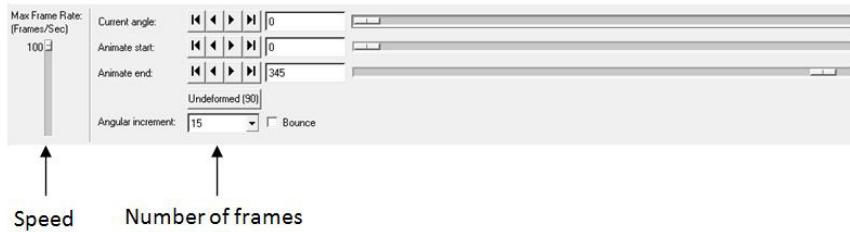
1. Load the .h3d file into HyperView using Load model
2. Use the Shaded Elements and Mesh lines button to turn the mesh on
3. Use the Contour button and plot Strain Energy
4. Change the traffic light from transient to linear/modal
5. Animate the first mode by clicking the Play button
6. If the deformation is too large change it to Model percent (10%) using the Deformed options
7. Add more frames and control the speed of the animation using the Animation controls button
8. Step through the modes using load case selector



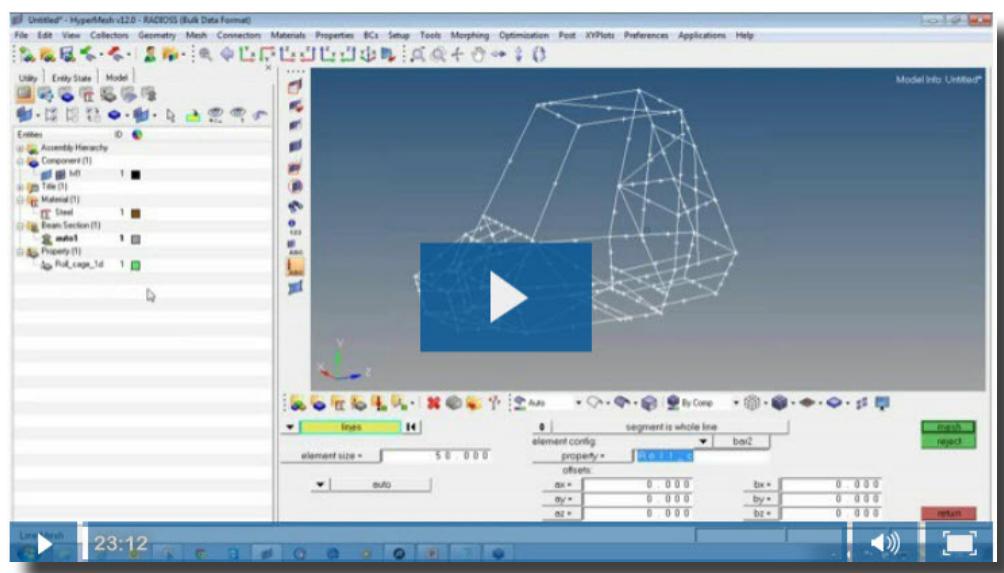
Deformed Panel (6):



Animation controls panel (7):



Videos / Webinars



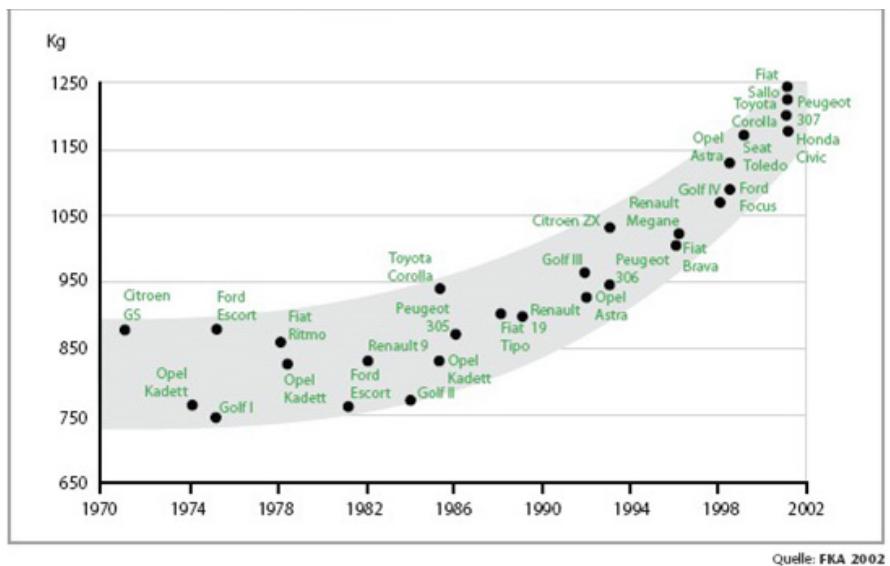
<https://altair-2.wistia.com/medias/lr7un6kib7>

18 Linear Buckling Analysis

This chapter includes material written by Markus Kriesch and André Wehr (Universität der Bundeswehr München) as well as material from Matthias Goelke.

18.1 Introduction

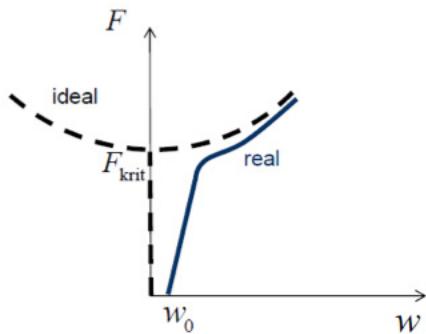
The demand for automobiles with less fuel consumption and less emissions has increased since the year 2000 because of rising commodity prices. However, the quest for light weight designs is (partly) counterbalanced by the additional weight related to passive and active safety systems and the steadily increasing comfort level for passengers. This creates a demand for intelligent lightweight concepts in many industries, not only in aerospace and automotive.



Besides the lightweight design, the increasing usage of optimization software leads to thin-walled and slender components which tend to buckle under axial loading.

Thin structures subject to compression loads that haven't achieved the material strength limits can show a failure mode called buckling. Buckling is characterized by a sudden failure of a structural member subjected to high compressive stress, where the actual compressive stress at the point of failure is less than the ultimate compressive stresses that the material is capable of withstanding (may be helpful is the explanation on Wikipedia (if still alive) <http://en.wikipedia.org/wiki/Buckling>)

In other words, once a critical load is reached, the slender component (for example a beam where its length is much larger than its cross sectional area) draws aside instead of taking up additional load.

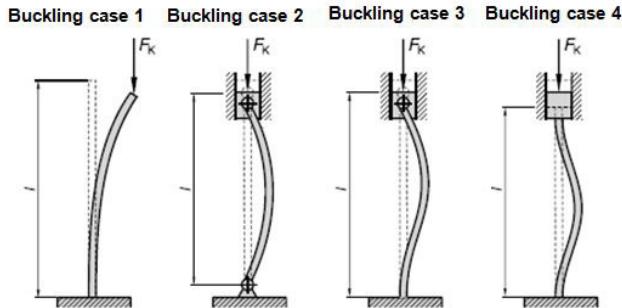


Force versus displacements of a slender beam; from Ihlenburg: Skript Technische Mechanik 2.7 Knicken. HAW Hamburg – FB MP; 2012

This failure can be analyzed using a technique well known as linear buckling analysis. The goal of this analysis is to determine the buckling load factor, λ , and the critical buckling load.

In OptiStruct, if the load factor λ is > 1 , the component is considered to be safe (i.e. the actual load can be multiplied by λ until buckling would occur).

18.2 Elastic Buckling



Euler buckling cases; K=effective buckling length factor; L=unsupported length of column
(from Läpple, Volker: Einführung in die Festigkeitslehre; Vieweg+Teubner Verlag; 2011)

In 1757 Leonhard Euler derived the following equation:

$$F_{\text{crit}} = \pi^2 E I / (K L)^2 = \pi^2 E I / s^2$$

where

F = maximum or critical force (vertical load on column)

E = modulus of elasticity; I = area moment of inertia (second moment of area)

L = unsupported length of column

K = column effective length factor, whose value depends on the conditions of the end support of the column, as follows:

- For both ends pinned (hinged, free to rotate), $K = 1.0$
- For both ends fixed, $K = 0.50$
- For one end fixed and the other end pinned, $K = 0.707$
- For one end fixed and the other end free to move laterally, $K = 2.0$

KL = the effective buckling length of the column

In other words, the critical force depends on:

- Length of column
- Cross-section (second moment of surface area)
- Material property (Young's modulus, in case of elastic material)
- Boundary condition (The boundary conditions determine the mode of bending and the distance between inflection points on the deflected column. The closer together the inflection points are, the higher the resulting capacity of the column.)

The strength of a column may therefore be increased by distributing the material so as to increase the moment of inertia. This can be done without increasing the weight of the column by distributing the material as far from the principal axis of the cross section as possible, while keeping the material thick enough to prevent local buckling. This bears out the well-known fact that a tubular section is much more efficient than a solid section for column service.

Note: Real constructions often contain imperfections, such as pre-deformations, due to which large displacements or failure may

occur even if the loading is still below the ideally critical load. The linear buckling analysis in general overestimates the strength/stability of the structure and leads to non-conservative results. Thus, it shouldn't be used as the only measure. However, the linear buckling analysis at least provides information about the expected deformation shapes..

Looking at Euler's equation and dividing it by the area A defines the critical stress at which buckling will occur:

$$\sigma_{\text{krit}} = F_{\text{crit}} / A \leq R_e$$

where R_e is the elastic limit.

$$\sigma_{\text{krit}} = F_{\text{crit}} / A = \pi^2 E I / A s^2 = \pi^2 E / \lambda^2 \leq R_e$$

with

$$\lambda = s / \sqrt{(I / A)}$$

18.3 Linear Buckling Analysis With OptiStruct

The problem of linear buckling in finite element analysis is solved by first applying a reference level of loading, F_{ref} , to the structure. This is ideally a unit load, F , that is applied. The unit load and respective constraints, SPC, are referenced in the first load steps/subcase. A standard linear static analysis is then carried out to obtain stresses which are needed to form the geometric stiffness matrix K_G . The buckling loads are then calculated as part of the second loadsteps/subcase, by solving an eigenvalue problem:

$$(K - \lambda K_G)x = 0$$

K is the stiffness matrix of the structure and λ is the multiplier to the reference load. The solution of the eigenvalue problem generally yields n eigenvalues λ , (buckling load factor) where n is the number of degrees of freedom (in practice, only a subset of eigenvalues is usually calculated). The vector x is the eigenvector corresponding to the eigenvalue.

The eigenvalue problem is solved using a matrix method called the Lanczos method. Not all eigenvalues are required. Only a small number of the lowest eigenvalues are normally calculated for buckling analysis.

The lowest eigenvalue is associated with buckling. The critical or buckling load is:

$$F_{\text{crit}} = \lambda_{\text{crit}} F_{\text{ref}}$$

In other words,

$$\lambda_{\text{crit}} = F_{\text{crit}} / F_{\text{ref}}$$

thus

$$\lambda_c < 1 \text{ buckling}$$

$$\lambda_c > 1 \text{ safe}$$

Note: The displacement results obtained with a buckling analysis depict the buckling mode shape. Any displacement values are meaningless. The same holds true for stress and strain results from a buckling analysis.

From Theory To Practice: How To Set Up A Linear Buckling Analysis

In order to run a linear buckling analysis, the following two steps are required:

Step 1

3 load collectors and 2 loadsteps/subcases must be specified:

- One Load collector for constraints (SPC = SinglePoint Constraints; no Card Image needed)
- One Load collector for loads (ideally unit load); no Card Image needed
- One load collector (with Card Image EIGRL) which defines the number of buckling modes to be determined (card image shown below)

- ND = 1: This tells OptiStruct to extract the first buckling mode



Step 2

Defining of 2 loadsteps; one for static and for buckling. The load collectors with the SPCs and the unit load define a static load subcase. Notice that the type is set to linear static:



Then the buckling load step (which needs the results from the static loadstep) is defined:



As before SPC references the load collector which contains the model constraints. STATSUB(BUCKLING) is referencing the static subcase/loadstep from before, and finally, METHOD references the load collector which bears the information about the number of eigen (buckling) modes to be extracted (i.e. load collector with Card Image EIGRL). Also notice that the type is now set to= linear buckling.

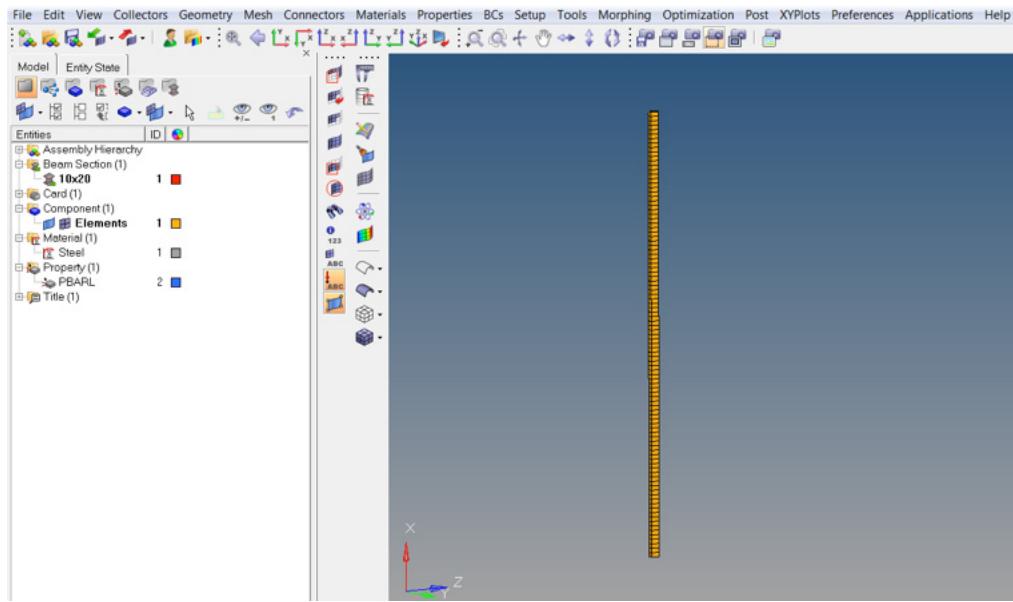
Note:

- STATSUB cannot refer to a subcase that uses inertia relief.
- The buckling analysis will ignore zero-dimensional elements, MPC, RBE3, and CBUSH elements. These elements can be used in buckling analysis, but they do not contribute to the geometric stiffness matrix, K_G .
- By default, the contribution from the rigid elements to the geometric stiffness matrix is not included. Users have to add PARAM, KGRGD, YES to the bulk data section to include the contribution of rigid elements to the geometric stiffness matrix.
- In addition, through the EXCLUDE subcase information entry, users may decide to omit the contribution of other elements to the geometric stiffness matrix, effectively allowing users to control which parts of the structure are analyzed for buckling. The excluded properties are only removed from the geometric stiffness matrix, resulting in a buckling analysis with elastic boundary conditions. This means that the excluded properties may still be showing movement in the buckling mode.

18.4 Example: Linear Buckling Analysis Of A Beam Structure

Step 1

Launch HyperMesh, select the OptiStruct User Profile and retrieve the file BAR_EULER_1.hm:



Imported model “BAR_Euler_1.hm”. The 1D elements are displayed via 3D Element Representation,

Model information:

$$b = 10 \text{ mm}$$

$$h = 20 \text{ mm}$$

$$l = 1000 \text{ mm}$$

$$E = 210.000 \text{ N/mm}^2, v=0.3 \text{ (steel)}$$

$$\text{Area moment of inertia} = 1.666,667 \text{ mm}^4$$

$$\text{Analytical critical buckling force } F_{\text{crit}} = 863,59 \text{ N (Euler case 1)}$$

The model is built on 1D elements (of type CBAR) with Property Card Image PBARL (in case you are interested to learn more about 1D elements, please see the corresponding chapter in this book). The material parameters correspond to steel.

In order to better understand the model details, please review the Card Images of the property and material collector, respectively.

Step 2

Create three load collectors.

- SPC (Single Point Constraints; no Card Image)
- Force (static unit load; no Card Image)
- Buckling (with Card Image EIGRL)

The load collectors can be easily created through right-click in the Model Browser window, and activate the option Create > Load Collector.

Then simply assign names and colors or your choice, respectively.

Name	Value
Name	SPC
ID	1
Color	
Include File	[Master Model]
Card Image	<None>

Name	Value
Name	force
ID	1
Color	
Include File	[Master Model]
Card Image	<None>

Creation of load collectors SPC and force (no Card image needed)

Name	Value
Solver Keyword	EIGRL
Name	buckling
ID	1
Color	
Include File	[Master Model]
Card Image	EIGRL
User Comments	Hide In Menu/Export
V1	
V2	
ND	1
MSG_LVL	
MAXSET	
SHFSCL	

Creation of the load collector named “buckling” with Card Image EIGRL

Clicking “Create” will bring up the Card Editor for this load collector:

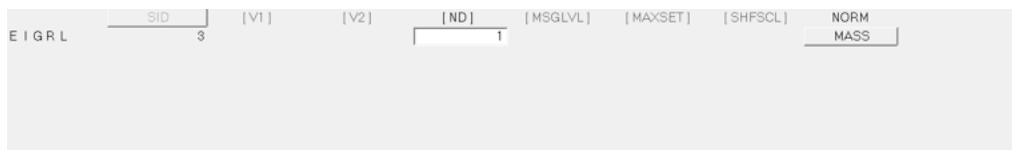
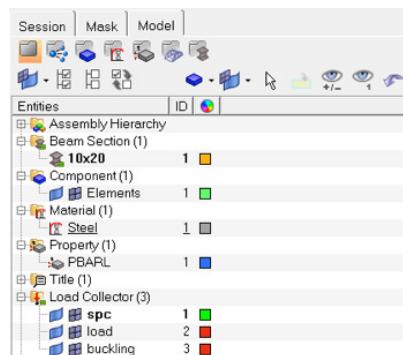


Figure: Card Image EIGRL

In the Card Image, activate ND and enter in a value of 1. This tells OptiStruct to extract the first buckling modes.

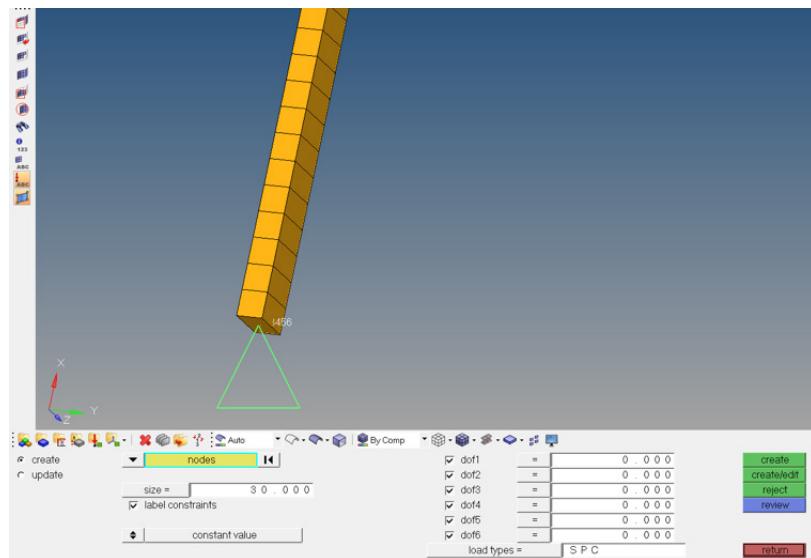
Step 3

Create the loads and boundary conditions. Make sure that the respective load collector is set to be the current collector (in the Model Browser right-click on, for instance SPC and select “Make Current”).



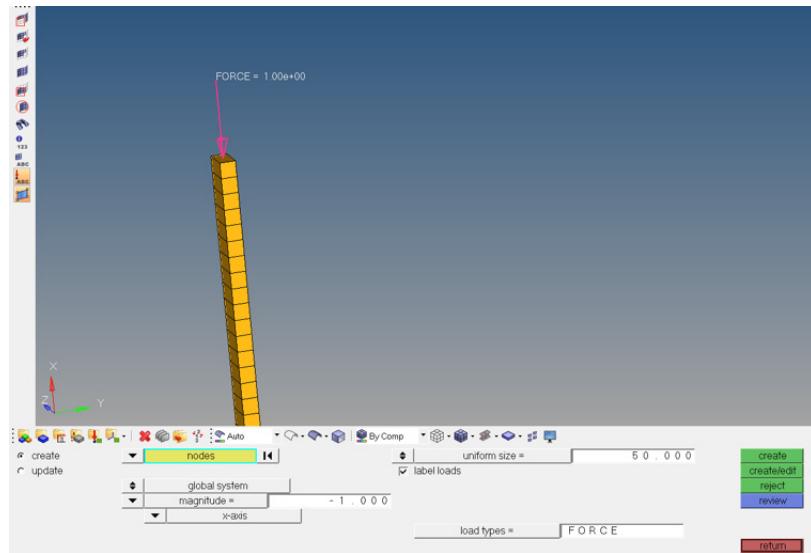
“Model Browser”. The load collector names SPC is set to be the current collector and hence listed in bold letters

In the constraints panel (BCs > Create > Constraints) select the node on the bottom face of the beam, as shown in the image below. All degrees of freedom dof are constraint (dof 1-6= 0).



Constraint panel. The node at the lower end of the column is constraint with respect to all of its dof's

Next, set the load collector force as the current collector and create the reference load (unit load). In the forces panel (BCs > Create > Forces) the node at the top of the beam is selected, as shown in the figure below:



Force Panel. A reference unit load in negative x-direction is applied (magnitude= -1, x-axis)

Step 4

Create the loadsteps. For the linear buckling analysis, two loadsteps/subcases are required.

- Linear static loading
- Linear buckling analysis

The subcases are created in the Loadsteps panel which can be accessed from the menu bar by selecting Setup > Create > LoadSteps.

Make sure the type is set to linear static. In here SPC references the single point constraints load collector (named SPC), and LOAD the load collector with the reference load (load):



Loadsteps panel (definition of a linear static subcase)

In the second loadstep, the buckling loadstep, make sure the type is set to linear buckling:



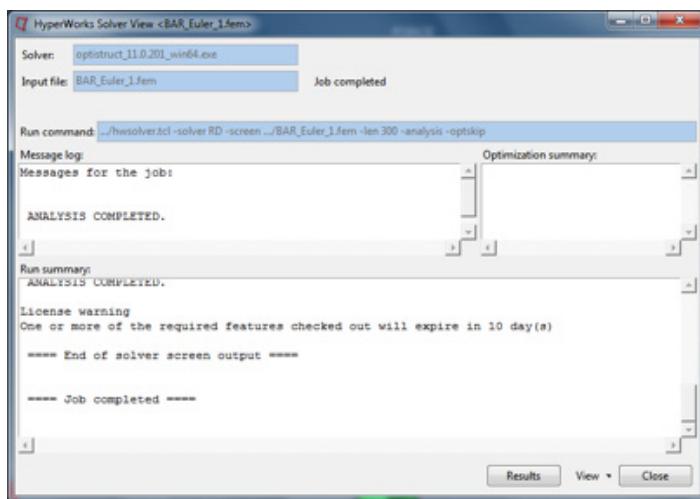
Loadstep panel (Definition of the linear buckling subcase) STATSUB references the static subcase and METHOD the load collector with the information about the buckling modes to be determined

That's all it takes – time to run the analysis.

Step 5

Start the computation run of OptiStruct. On the Analysis page, select the OptiStruct panel, chose the working directory and then save the model as *.fem (the extension *.fem indicates in input files for OptiStruct). Finally, click OptiStruct to execute the calculation.

With the start of the calculation, the Solver View appears. This shows the steps of the calculation and possible errors in the Message log area.



HyperWorks Solver View

Note: the *.out file includes information from the model check run before the calculation as well as additional base information of the calculated results. Also, check this file error messages and warnings that will help you debug your input deck if any errors are present. In general, always view this file, even if the analysis run was successful.

Step 6

Postprocessing of the analysis. If the job is successful, you should see new results files in the working directory. The working directory is the directory where the model was saved.

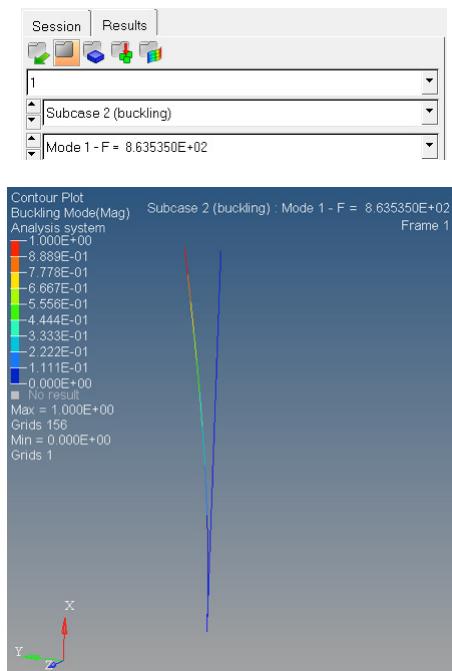
The default files that will be written to your directory are:

File	Explanation
*.h3d	Binary HyperView result file
*.res	Includes all the results of displacement through stress, which can be shown in the Post panel of HyperMesh
*.out	ACSI-based output file, which includes information about the model check run before the calculation and additional base information of calculation results
*.stat	Detailed breakdown of the elapsed CPU-time for all important analysis steps

Table: Standard computed files

To view the results, start **HyperView**.

Make sure and pay attention to the loadstep/subcase selected for display. This is controlled in the Results Browser.



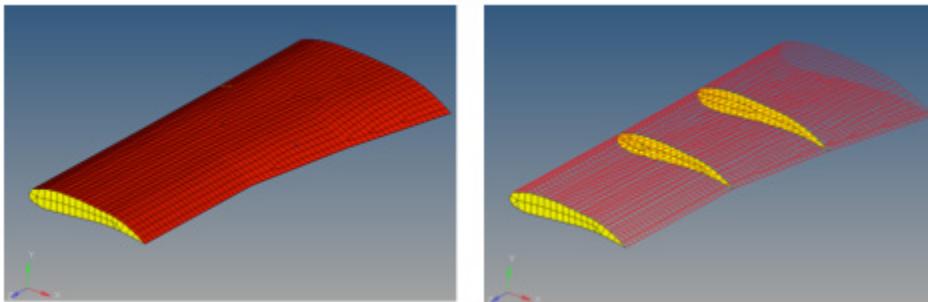
Subcase 2 “buckling” in HyperView. Reference (undeformed) model in blue

The critical buckling load for the calculated bar (with unit load 1 N) is $F = 863,53$ N (critical buckling factor $\lambda_{cr} = 863.53$). In other words, the model could be loaded 863 times the unit load until buckling would occur. Of course, we need to remember that this is an ideally linear buckling analysis and that real parts tend to buckle at lower loads than the theoretical value due to imperfections.

18.5 Example: Wing Linear Buckling Analysis

This exercise runs a linear buckling analysis on a simple aircraft wing. This is a typical problem for aerospace structures that need to be very light and consequently become slender. Because the structure has a high slenderness ratio, the buckling failure verification becomes necessary. The objective of this project is to verify if the 3 static load cases applied to the wing will not make it fail.

In this exercise, you will learn how to verify a wing baseline design for buckling criteria.



Model Information (Wing.hm):

Design Criteria:

Buckling: first mode > (1.5 x).

(Static: U < 20 mm and Von Mises < 70 MPa)

Material Aluminum:

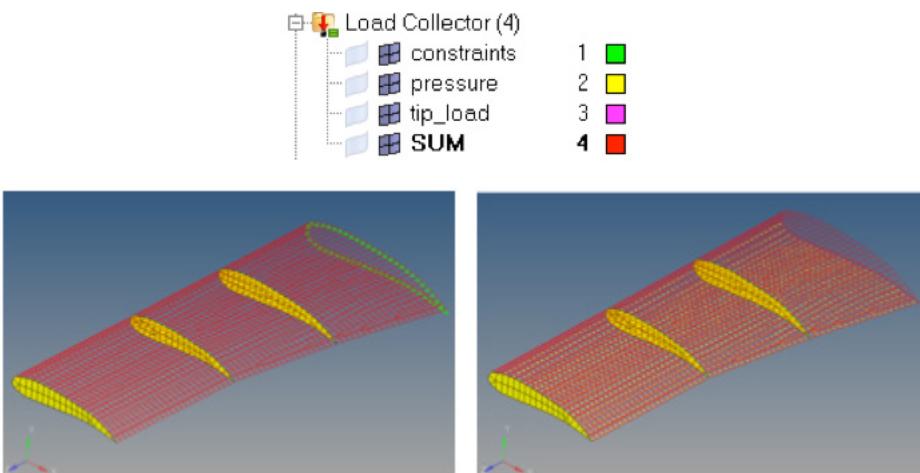
$$\rho = 2.1 \times 10^{-9} \text{ T/mm}^3 \text{ [RHO] Density}$$

$$E = 70.000 \text{ MPa} \text{ [E] Young's modulus}$$

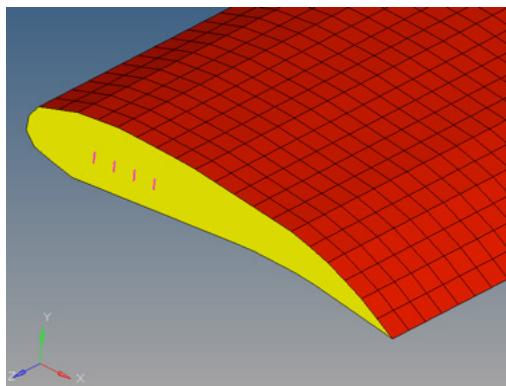
$$\nu = 0.33 \text{ - [nu] Poisson's ratio}$$

In the given model (Wing.hm), the load, constraints and loadcases have already been created. Please review the property, material, and load collectors before starting the baseline analysis.

Here, we focus on the load collectors:



Left: Applied constraints (green triangles) are in the load collector constraints. Right: Applied pressure (yellow arrows) are in the load collector pressure.



Tip load at the wing (purple arrows are in the load collector tip_load)

In the load collector named SUM (with Card Image LOAD), the loads from the load collectors pressure (ID=2) and tip_load (ID=3) are combined. S, S1(1) ,and S1(2) are weighting factors, respectively.

LOAD	ID	S	S1(1)	L1(1)	S1(2)	L1(2)	
	4	1 . 0 0 0	1 . 0 0 0		2	1 . 0 0 0	3

User Comments
▼ Hide In Menu/Export
LOAD_Num_Set = 2

reject
default

abort
return

The three static loadsteps are:

Load Step (3)

- PRESSURE 1
- TIP 2
- SUM 3

P R E S S U R E

name: P R E S S U R E	type: linear static	create
SPC	MPC	edit
LOAD	TEMP	update
SUPPORT1	DEFORM	review
PRETENSION	STATSUB(PRELOAD)	
	STATSUB(PRETENS)	

T I P

name: T I P	type: linear static	create
SPC	MPC	edit
LOAD	TEMP	update
SUPPORT1	DEFORM	review
PRETENSION	STATSUB(PRELOAD)	
	STATSUB(PRETENS)	

S U M

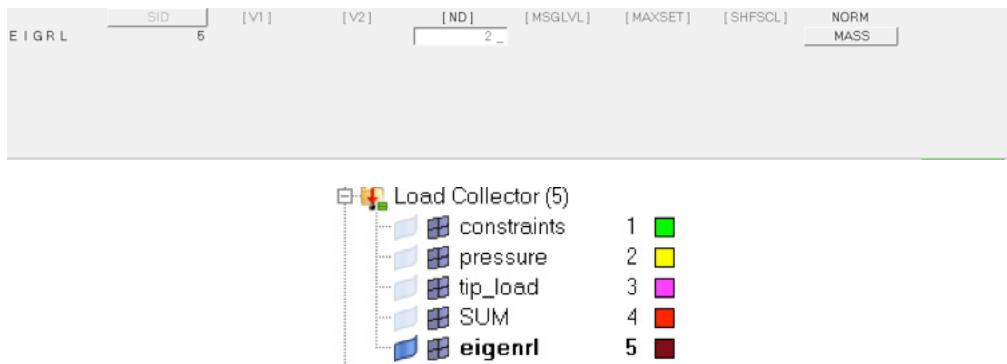
name: S U M	type: linear static	create
SPC	MPC	edit
LOAD	TEMP	update
SUPPORT1	DEFORM	review
PRETENSION	STATSUB(PRELOAD)	
	STATSUB(PRETENS)	

Note, the analysis type is set to linear static, respectively.

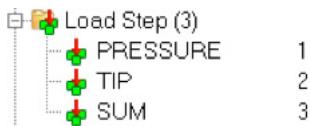
In order to define the buckling load step, another load collector is needed. Do you recall what the missing load collector is about?

We need to create a load collector with Card Image EIGRL (to define the number of buckling modes we are interested in).

It's Card Image is



With respect to the loadsteps of interest,



we define the buckling loadsteps next.

Please compare the ID's of the referenced load collectors (SPC= and METHOD =) and loadstep (STATSUB =)

<input type="text" value="buck_pressur"/>	<input type="text" value="linear buckling"/>	<input type="button" value="create"/>	
<input checked="" type="checkbox"/> SPC	<input type="text" value="1"/>	<input checked="" type="checkbox"/> METHOD(STRUCT)	<input type="text" value="5"/>
<input type="checkbox"/> MPC		<input type="checkbox"/> DEFORM	
<input checked="" type="checkbox"/> STATSUB(BUCKLING)	<input type="text" value="1"/>	<input type="button" value="edit"/>	
<input type="button" value="update"/>			
<input type="button" value="review"/>			
<input type="button" value="return"/>			
<input type="text" value="buck_tip"/>	<input type="text" value="linear buckling"/>	<input type="button" value="create"/>	
<input checked="" type="checkbox"/> SPC	<input type="text" value="1"/>	<input checked="" type="checkbox"/> METHOD(STRUCT)	<input type="text" value="5"/>
<input type="checkbox"/> MPC		<input type="checkbox"/> DEFORM	
<input checked="" type="checkbox"/> STATSUB(BUCKLING)	<input type="text" value="2"/>	<input type="button" value="edit"/>	
<input type="button" value="update"/>			
<input type="button" value="review"/>			
<input type="button" value="return"/>			
<input type="text" value="buck_sum"/>	<input type="text" value="linear buckling"/>	<input type="button" value="create"/>	
<input checked="" type="checkbox"/> SPC	<input type="text" value="1"/>	<input checked="" type="checkbox"/> METHOD(STRUCT)	<input type="text" value="5"/>
<input type="checkbox"/> MPC		<input type="checkbox"/> DEFORM	
<input checked="" type="checkbox"/> STATSUB(BUCKLING)	<input type="text" value="3"/>	<input type="button" value="edit"/>	
<input type="button" value="update"/>			
<input type="button" value="review"/>			
<input type="button" value="return"/>			

Finally, run the analysis and view the buckling modes.

18.6 Example: Buckling With Gravitational Load

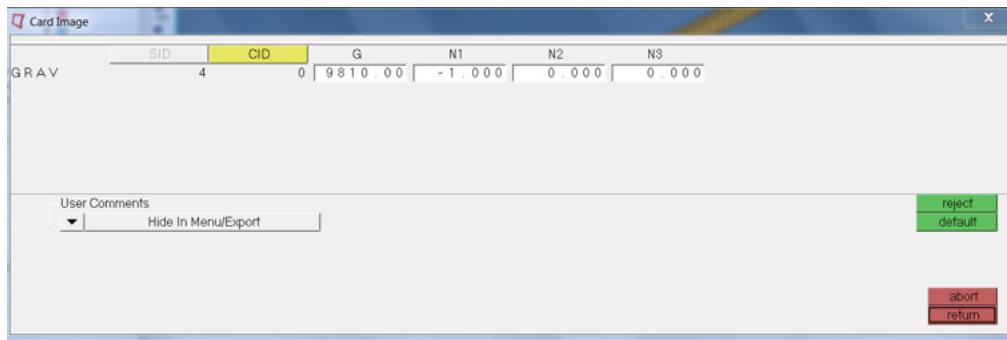
Typically, the critical load factor is determined with respect to all loads defined in the referenced linear static loadcase (referenced by STATSUB).

In case the critical load factor should only be applied to variable loads (e.g. pressure, forces) and NOT to the gravitational load (constant load), the two procedures/techniques described below may be employed. The below described working steps are considered in the model file: Prestressed_gravity.zip.

Working Procedure A

In case gravitational and variable loads are to be considered, the following load collectors are required:

- Load collector for variable loads (here named force; no Card Image needed)
- Load collector for gravity (Card Image GRAV; remind the system units)



Load collector which combines variable and constant loads (Card image LOAD needed; here named: gravity_and_force).

The screenshot shows a 'Load Collector (8)' tree view and a 'Card Image' window for a 'LOAD' card.

Load Collector (8) Tree:

- EIGRL 3 (blue)
- spc 1 (green)
- force 2 (pink)
- gravity 4 (blue)
- gravity_and_force 5 (brown)** (highlighted)

Card Image Window (LOAD):

ID	S	S1(1)	L1(1)	S1(2)	L1(2)
LOAD	6	1 . 0 0 0 1 . 0 0 0	4	1 . 0 0 0	2

User Comments: Hide In Menu/Export

reject default abort return

In the Card Image above, L1(1) is linked to the gravity load collector, L1(2) refers to the variable load (in this case, force). S, S1(1) and S1(2) are scale factors, respectively.

In addition, we have a load collector for the model constraints (here named spc; no Card image needed) and a load collector which defines the number of eigenmodes to be extracted (Card Image EIGRL, here the load collector is named EIGRL).

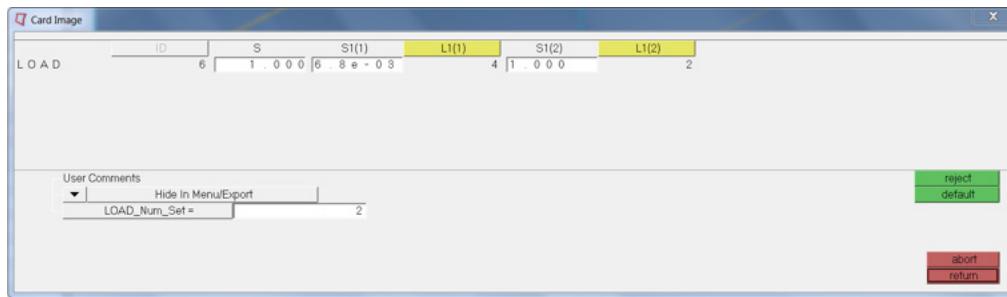
Step 1:

Determine the critical load factor for all loads (constant and variable loads). We may call this critical load factor buckling_all (i.e. run the buckling analysis taking into account all loads).

Step 2:

Scale the constant load, L1(1), by adjusting the associated scale value S1(1) in the Card Image of the load collector:

$$S1(1) = 1 / [\text{critical load factor "buckling_all"}]$$



The scale factor S1(2) of the variable load L1(2) remains unchanged i.e. S1(2)=1.0.

Step 3:

Run the buckling analysis with the newly scaled value for the gravity load, L1(1). We call this additional buckling analysis and critical load factor, A_buckl_grav_force.

Provided that the buckling factor A_buckl_grav_force equals buckling_all, everything is fine, because then:

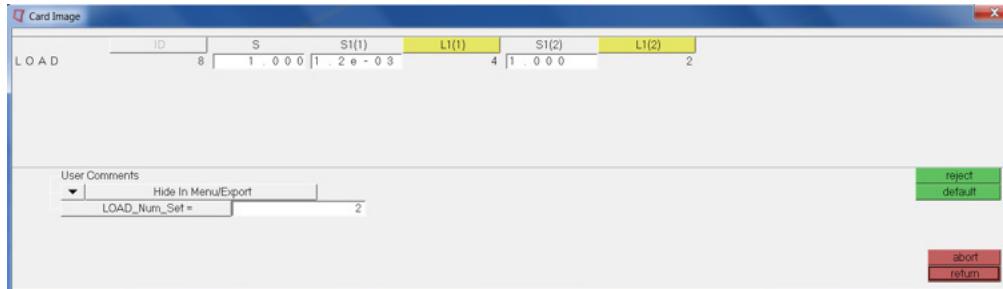
gravitational load L1(1) = L1(1) / [critical load factor buckling_all] * [critical load factor A_buckl_grav_force]

However, if A_buckl_grav_force ≠ buckling_all then Step2 to Step 3 must be repeated iteratively.

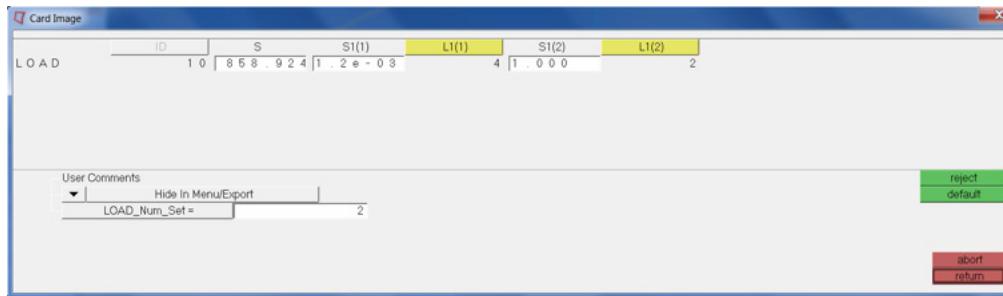
Now the gravity load L1(1) is scaled by A_buckl_grav_force:

$S1(1) = 1 / [A_{buckl_grav_force}]$

This will lead to another buckling value which may be called B_buckl_grav_force and so on.



Eventually, when the new (e.g. C_buckl ...) and the last (e.g. B_buckl ...) buckling factors are equal, an additional review buckling analysis can be carried out. In this analysis, ALL loads are scaled by the last buckling factor e.g. C_buckl ...



This buckling analysis will then depict a buckling factor very close to ±1.

Working Procedure B

Starting with the OptiStruct solver update HW11.0.220-HW (July 2012) the above described process becomes much easier and more straight forward.

In addition to the load collectors spc" and EIGRL (see above), we now need the two load collectors:

- Load collector for variable loads (here named force; no Card Image needed)
- Load collector for gravity (Card Image GRAV; remind the system units)



Based on these two load collectors, the two loadcases are defined:

- Loadcase forces_only



- Loadcase gravity_only



Note: Currently (July 2012) the definition of a buckling loadstep in which the gravitational/constant loads are neglected requires you to select type: generic.

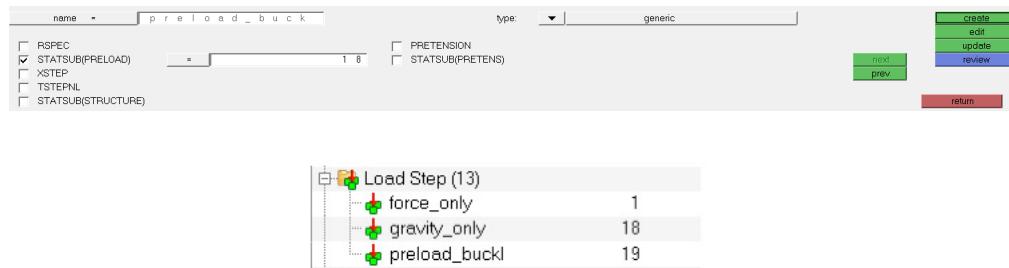
As before SPC references the model constraints, METHOD (STRUCT) references the load collector with the Card Image EIGRL,



STATSUB (BUCKLING) references forces_only, and



eventually, the gravity loading gravity_only is applied as a pre-load.



You can view the load collectors and loadsteps in the file: prestressed_gravity.hm

18.7 Linear Buckling Analysis Tutorials And Videos

Recommended Tutorials:

The following tutorial is part of the HyperWorks Help Documentation:

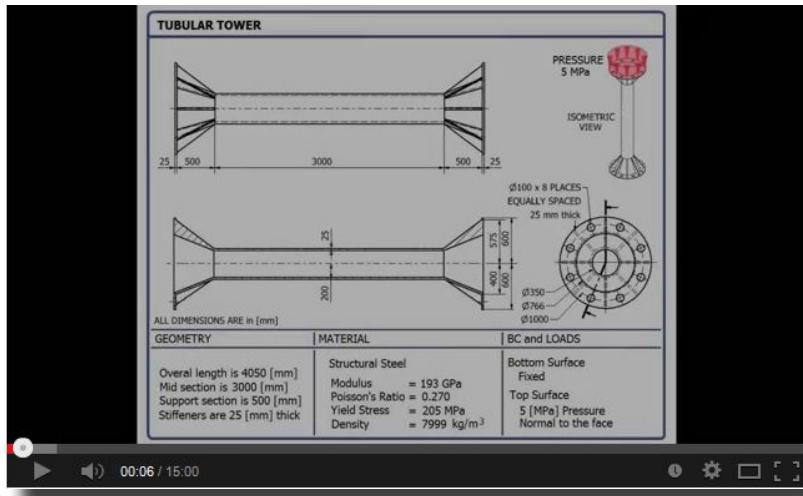
- OS-1040: 3D Buckling Analysis using OptiStruct

Demo Files:

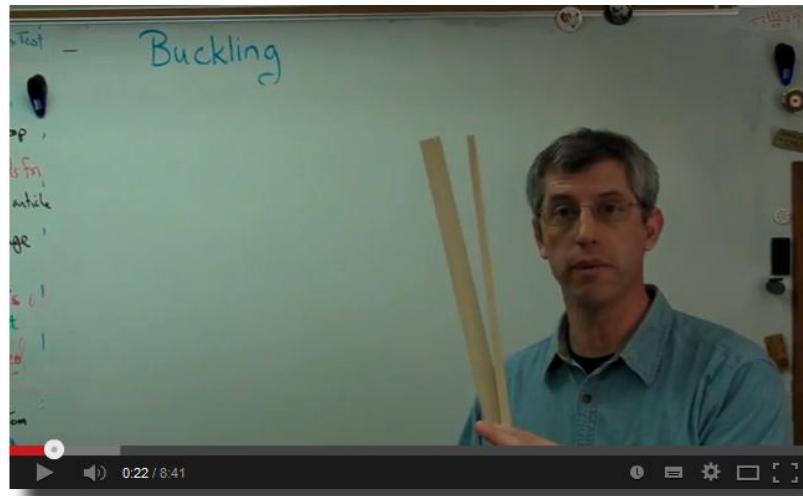
- Buckling analysis of a bottle
- Buckling analysis of the central wing of aircraft
- MBD Cable showing torsional buckling

Videos On Youtube

- HyperMesh 12 - Tower – Buckling (<http://youtu.be/BueeQiwcOJY>, 15 minutes)



- Buckling of a Thin Column (<http://youtu.be/wrd08hPJGyg>, 8 minutes)



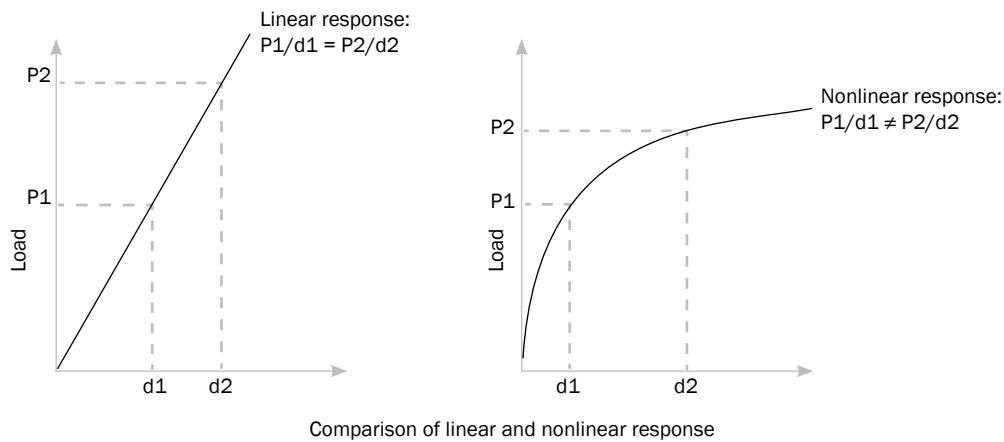
19 Nonlinear Analysis

This chapter includes material from the book “**Practical Finite Element Analysis**” and the HyperWorks Help Documentation. Additional material was also added by Patrick Zerbe, Kristian Holm, Christian Steenbock and Matthias Goelke.

19.1 Introduction

In this chapter, we discuss the practical aspects of nonlinear static Finite Element Analysis.

But how do we know that our problem is nonlinear? The best way is to look at the load-displacement response of one or more characteristic load introduction points. As discussed in an earlier chapter on Types of Analysis, when the structural response (deformation, stress and strain) is linearly proportional to the magnitude of the load (force, pressure, moment, torque, temperature etc.), then the analysis of such a structure is known as linear analysis. When the load to response relationship is not linearly proportional, then the analysis falls under nonlinear analysis (see figure below). For example, when a compact structure made of stiff metal is subjected to a load relatively lower in magnitude as compared to the strength of the material, the deformation in the structure will be linearly proportional to the load and the structure is known to have been subjected to linear static deformation. But most of the time either the material behavior is not linear in the operating conditions or the geometry of the structure itself keeps it from responding linearly. Due to the cost or weight advantage of nonmetals (polymers, woods, composites etc.) over metals, nonmetals are replacing metals for a variety of applications. These applications have nonlinear load to response characteristics, even under mild loading conditions. Also the structures are optimized to make most of its strength, pushing the load level so close to the strength of the material, that it starts behaving nonlinearly. In order to accurately predict the strength of the structures in these circumstances, it is necessary to perform a nonlinear analysis.



As discussed in the earlier chapter, the stiffness matrix relating to the load and response is assumed to be constant for static analysis; however, all the real world structures behave nonlinearly. The stiffness matrix consists of geometric parameters such as length, cross sectional area and moment of inertia, etc. and material properties such as elastic modulus, rigidity modulus etc. The static analysis assumes that these parameters do not change when the structure is loaded. On the other hand, nonlinear static analysis takes into account the changes in these parameters as the load is applied to the structure. These changes are accommodated in the analysis by rebuilding the stiffness matrix with respect to the deformed structure (i.e. altered properties) after each incremental load application. Although, the world is nonlinear, it should be mentioned that in many cases the assumption of linearity is valid and in that way a linear analysis can be done instead. Also, from a computation point of view, it is a much less expensive approach.

19.2 Comparison Of Linear And Nonlinear FEA

The table below provides a short summary overview about the multiple differences between a linear and nonlinear FE analysis and is not meant to explain the differences in detail. Some of the main differences such as load-displacement relation, stress-strain relation, stress-strain measures are explained in some more detail in the remainder of this chapter.

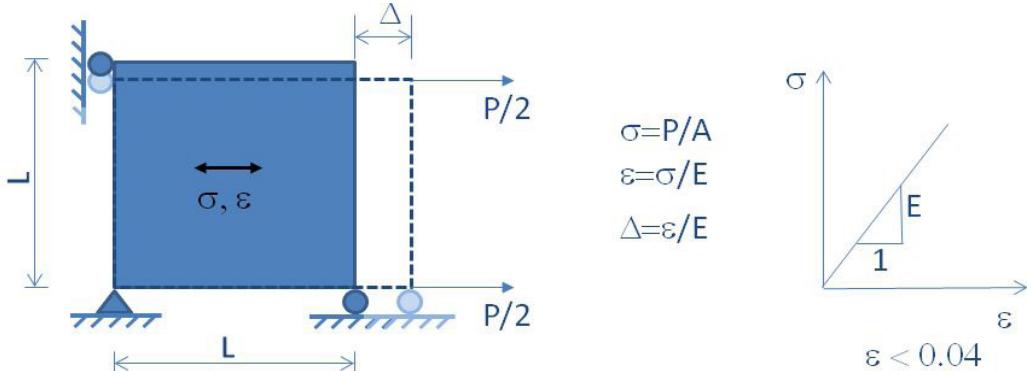
Characteristic Feature	Linear Problems	Nonlinear Problems
Load - Displacement Relation	Displacements vary linearly with applied loads. Thus Stiffness is constant. Changes in geometry due to displacement are assumed to be small and hence are ignored. Original or undeformed state is always used as a reference state.	It is nonlinear. Thus stiffness varies as a function of load. Displacements can be very large and changes in geometry cannot be ignored. Thus stiffness varies as a function of load.
Stress-Strain Relation	Linear up to the proportional / elastic limit. Properties such as Young's Modulus are easily available.	It is nonlinear function of stress-strain and time. These are difficult to obtain and requires lot of additional experimental material testing. Mind the differences between true stress and engineering stress
Superposition	Applicable. That is if a 1 N force causes x units of displacement, then 10 N magnitude of force will cause $10x$ displacement	Not applicable.
Reversibility	The behaviour of the structure is fully reversible upon the removal of external loads. This also means that loading sequence is not important and the final state is unaffected by the load history.	The final state after removal of loads is different from the initial state. Due to this superposition of load cases is not possible. Load history is very important.
Solution scheme	The load is applied in one step with no iterations.	Load is split into small increments with iterations performed to ensure that equilibrium is satisfied at every load increment.
Computational Time	Small	Large
User's Interaction with the software	Least required	Requires lot of monitoring as the software may fail to converge sometimes.

19.3 Types Of Nonlinearity

There are three basic source of nonlinearity: Geometric Nonlinearity, Material Nonlinearity, and Contact Nonlinearity.

Recall the definition of linear elastic – displacements will be small and stress is proportional to strain

Linear Elastic



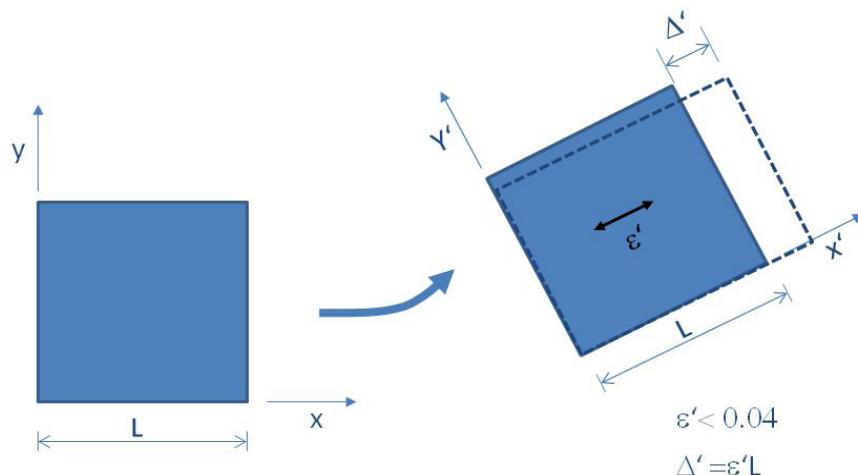
1) Geometric Nonlinearities

Geometric nonlinearity may be related to 1) Large strain 2) Large rotation, and 3) Large deformation

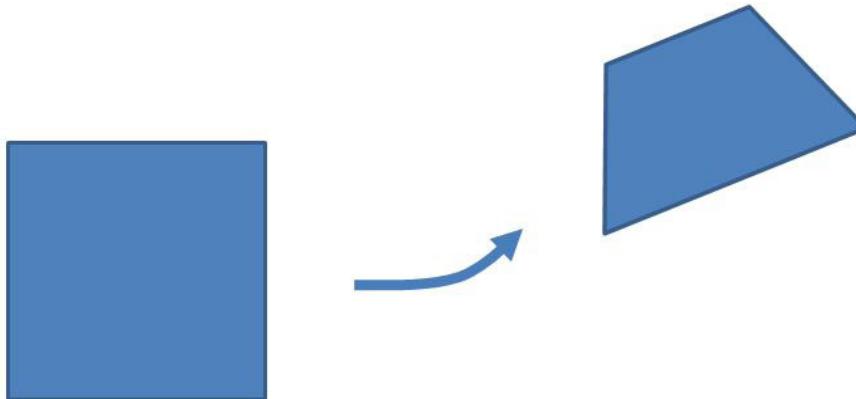
Geometric nonlinearity takes into account that the geometry cross section may change as a result of large deformation (in linear static analysis the cross section is considered to be constant). Large displacements may also be introduced by geometric buckling. A phenomena which is characterized by a sudden failure of a structural member subjected to high compressive stress, where the actual compressive stress at the point of failure is less than the ultimate compressive stresses that the material is capable

of withstanding. Thus, the equilibrium equations must be written with respect to the deformed structural geometry. Additionally, applied loads may change their direction as they increase, as when pressure inflates a membrane (Ref. Concepts and applications of finite element analysis; R.D. Cook et. al, p. 595).

Large Displacements And Rotations (small strain; linear or nonlinear material)



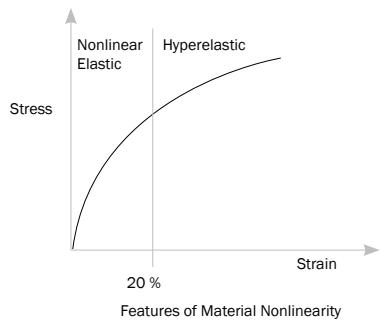
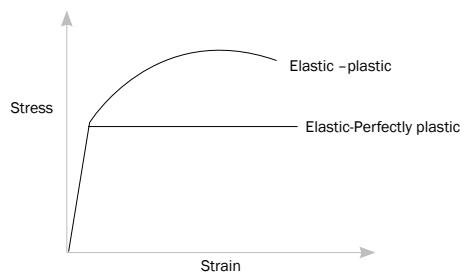
Large Displacements, Rotations, And Large Strain (linear or nonlinear material)



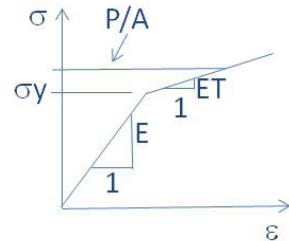
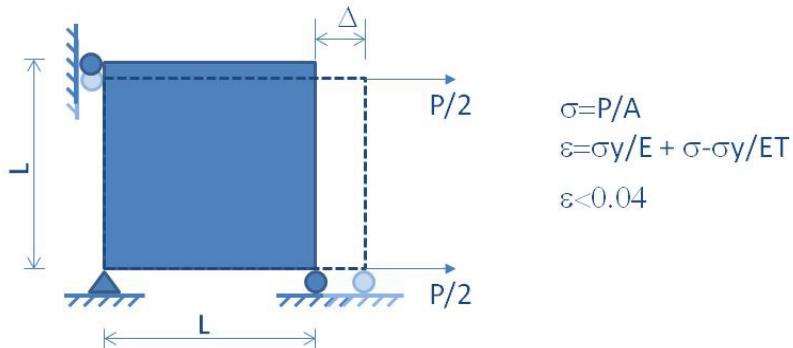
Images after K.J. Bathe, Finite Elemente Methoden

In linear FEA, strain, for example in the x -direction, is expressed as $\varepsilon_x = \partial u / \partial x$. Thus, in other words, only the first order term of $\varepsilon_x = \partial u / \partial x + \dots [(\partial u / \partial x)_z + (\partial v / \partial x)_z + (\partial w / \partial x)_z]$ is considered. In the case of large displacements (nonlinear), the second order terms are also considered. Additionally, that the stress-strain (material) relationship may or may not be linear.

2) Material Nonlinearities



Nonlinear Material (small displacements)



Images after K.J. Bathe, Finite Elemente Methoden

All engineering materials are inherently nonlinear as it is not feasible to characterize a nonlinear material by a single constitutive law for the entire range of environmental conditions such as loading, temperature and rate of deformation. We can idealize or simplify the material behavior to account for only certain effects which are important for the analysis. The linear elastic (also called Hookean) material assumption is the simplest of all. The material is nonlinear elastic if the deformation is recoverable and plastic if it is irrecoverable. If the temperature effects on the material properties are important, then the coupling between the mechanical and thermal behavior should be properly taken into consideration through thermo-elasticity and thermo-plasticity. If the strain rate has significant effects on the material, we have to consider the theories of visco-elasticity and visco-plasticity. An example of material nonlinear behavior has been given in the above figures.

A brief classification can be given as follows:

1. Nonlinear elastic
2. Hyperelastic
3. Elastic-perfectly plastic
4. Elastic-time independent plastic

5. Time dependent plastic (Creep)
6. Strain rate dependent elasticity – plasticity
7. Temperature dependent elasticity and plasticity

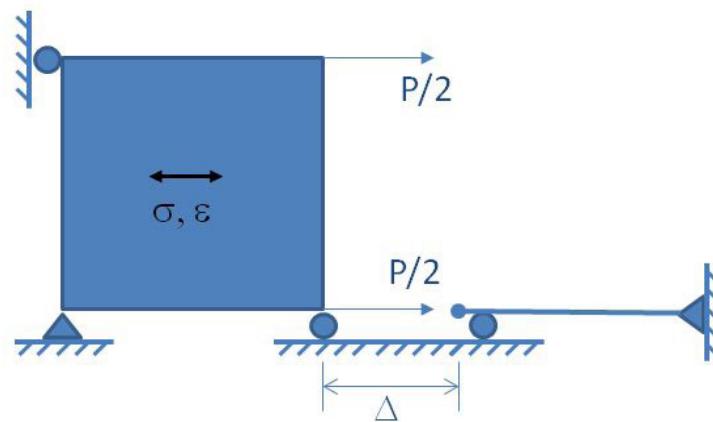
If we observe the stress-strain curves in the above figures, then material nonlinearity can be classified further as follows:

1. Linear elastic-perfectly plastic
2. Linear elastic-plastic. The plastic part in the stress-strain curve is time independent and can be analysed into two main types:
 - a. Elastic – Piecewise linear plastic
 - b. Elastic – Actual stress strain curve information.
3. Nonlinear elastic model characterizing materials with no fixed definition of yield point such as say plastic but the strain still limiting well below say 20 %.
4. Hyperelastic materials such as rubber undergoing very large displacements. A typical application is a gasket material.

3) Boundary Nonlinearities/Contact Nonlinearity

After displacements in the order of Δ have been reached (occurred), the boundary condition is changing, i.e. changes in/of the degree of freedom.

Contact



Images after K.J. Bathe, Finite Elemente Methoden

Boundary nonlinearity arises when boundary conditions in a FE model change during the course of the analysis. The boundary conditions could be added to or removed from the model due to boundary nonlinearity as the analysis progresses. This kind of nonlinearity typically involves contact sets in the model which could get engaged or disengaged as a response to applied loads. The load transfer mechanism via a contact pair is a complicated phenomenon. Researchers have developed several theories to describe the load transfer using contact sets. A typical contact algorithm in FE codes works as follows:

First, check for either open or closed contact surfaces. This is done by measuring the distances between the nodes on the surfaces forming the contact pairs. If a contact is closed, then apply reaction forces to the nodes in contact to keep them from penetrating through each other. If a contact is open, then no load is transferred between those nodes. The reaction forces applied to the nodes are calculated based on stiffness of the materials of each of the surfaces in the contact set. Since load transfer through a contact set is not constant and it requires continuous check to determine the open or closed status of the contact, this analysis can not be handled by linear static analysis codes. More accurate results can be obtained by checking the open or closed status of the nodes in the contact set more frequently.

Contact Treatment

There are two approaches to deal with contact:

- Penalty method
- Lagrange Multiplier method

The Penalty method is the most commonly used in explicit codes and can be found in most of OptiStruct interfaces. The Lagrange Multiplier method is used in special case studied.

Interfaces using the Penalty method are based on master / slave treatment. Contact can only occur between a set of slave nodes and a set of master segments. Master segments are defined depending on the type of element they lie on. If it is a 3-node or a 4-node shell, the segment is the surface of the element. If it is a solid element, the segment is defined as a face, Finally, if it is a 2D solid element (quad), the segment is a side.

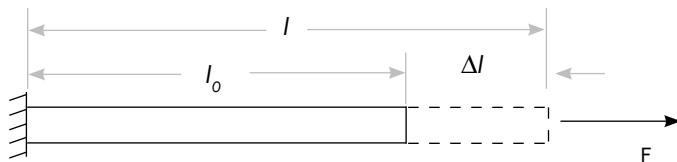
Contrary to the penalty method, the Lagrange Multiplier method is purely mathematic and does not require physical elements (springs) to model contact. A nonlinear system of equations is solved to account for contact conditions. Therefore, there is no time step collapse due to high interface stiffness, but it takes more CPU time to do 1 cycle as new equations need to be solved by nonlinear solver. The method has the advantage to stop slave nodes at the contact surface (that is, the contact condition is precisely satisfied); however the friction cannot be computed.

Additional information about this topic is available in the Chapter entitled “Contact Modelling in Structural Simulation – Approaches, Problems and Chances” written by Professor Rolf Steinbuch (Reutlingen University, Germany).

19.4 Stress-Strain Measures For Nonlinear Analysis

This is a major distinguishing factor between linear static and nonlinear FEA. In linear analysis we always use the engineering stress – strain definitions, whereas in nonlinear analysis typically the true stress and train is used. An excellent summary about stress/strain relationships is also available at <http://dolbow.cee.duke.edu/TENSILE/tutorial/node3.html>.

We will examine these various stress and strain definitions through a simple one-dimensional example.



A one dimensional rod undergoing axial deformation

1) Engineering Strain And Engineering Stress

Engineering strain is a small strain measure, which is computed using the original geometry. The engineering strain measure is a linear measure since it depends on the original geometry, (i.e. length) which is known beforehand. It is limited to small rotations of the material because a moderate rigid body rotation will result in non-zero strains.

$$\varepsilon_x = \frac{\Delta l}{l_0}$$

Engineering stress (σ), is the conjugate stress measure to engineering strain (ε). It uses the current force F and the original area A_0 in its computation.

$$\sigma = \frac{F}{A_0}$$

2) Logarithmic Strain And True Stress.

Logarithmic strain / natural strain / true strain is a large strain measure, which is computed as

$$\varepsilon_l = \int_{l_0}^l \frac{dl}{l_0} = \ln \left(\frac{l}{l_0} \right)$$

This measure is a nonlinear strain measure since it is a nonlinear function of the unknown final length. It is also referred to as the log strain (or true strain) and represents an additive strain measure as compared to linear strain. Let us consider a bar with say initial length of 1m undergoing deformation in 3 steps as follows:

Step 1 : deformation from 1 m to 1.2 m

Step 2 : deformation from 1.2 to 1.5 m

Step 3 : deformation from 1.5 to 2 m

In the following table we compare the engineering strains against true strains and one can clearly see that the additivity is retained in only true strain and hence this one should be used in nonlinear analysis.

Step	Engineering Strain	
1	0.2	Log (1.2/1) = 0.1823
2	0.3/1.2=0.25	Log (1.5/1.2) = 0.2231
3	0.5/1.5=0.33	Log (2/1.5) = 0.2876
Addition of strains in all the steps	0.78	0.6931
Total : From 1 to 3	(2.0 - 1.0)/1.0 = 1.0	Log(2/1) = 0.6931
Conclusion	Engg. Strain is not additive and doesn't preserve the history of deformation correctly .	True Strain is additive and thus preserves the history of deformation correctly.

True stress (s) is the conjugate 1-D stress measure to the log strain (ε_l), which is computed by dividing the force F by the current (or deformed) area A . This measure is also commonly referred to as the Cauchy stress.

$$s = \frac{F}{A}$$

3) Green-Lagrange Strain And 2nd Piola-Kirchoff Stress

Green-Lagrange strain is another large strain measure, which is computed in 1-D as

$$\varepsilon_G = \frac{1}{2} \left(\frac{l^2 - l_0^2}{l_0^2} \right)$$

This measure is nonlinear because it depends on the square of the updated length l , which is an unknown. A computational advantage of this strain measure, over the log or Hencky strain, is that it automatically accommodates arbitrarily large rotations in large strain problems. The conjugate stress measure for the Green-Lagrange strain, is the 2nd Piola-Kirchoff stress (s). It is computed in 1-D by

$$s = \frac{l_0}{l} \frac{F}{A_0}$$

It should be noted that this stress has little physical meaning.

19.5 Essential Steps To Start With Nonlinear FEA

- Learn first how the software works on a simple model before you use a nonlinear feature which you haven't used. Also guess how your structural component will behave, i.e. check for available studies, reports and benchmarks .
- Try to understand the software's supporting documentation, its output and warnings.

- Know what you are looking for. Prepare a list of questions you think your analysis should be able to answer. Design the analysis, including the model, material model, and boundary conditions, in order to answer the questions you have in mind.
- Keep the final model as simple as possible. A linear analysis done first can provide a lot of information such as where are the high stresses in the model, where the initial contact may occur, and what level of load will introduce plasticity in the model. The results of the linear analysis may even point out that there is no need for a nonlinear analysis. Examples of such a situation include the yield limit not being reached, there is no contact, and the displacements are small.
- Verify and validate the results of the nonlinear FEA solution. Verification means that “the model is computed correctly” from the numerical point of view. Wrong discretization with respect to the mesh size and time stepping are common errors. Validation asks the questions if “the correct model” is computed e.g. the geometry, material, boundary conditions, interactions etc coincide with the one acting in reality.
- Try to look into the assumptions made with respect to the structural component, its geometry behavior with respect to large strain (On/Off), look into different material models if the earlier model is unable to give you a result you expect (sometimes software only make some models compatible with commonly used elements and in this case you might look into a possibility of changing the element formulation).

19.6 General Procedure For Nonlinear Static Analysis Project

A typical nonlinear static analysis project requires following steps:

Meshing: FE model building is a very important step in FE analysis, irrespective of what kind of analysis is going to be performed. The selection of the appropriate element type for certain applications has already been discussed in Chapters 4-7. The FEA group is provided with a part with surface data, which is required to be meshed with elements to get the component mesh. When all the parts in the assembly are meshed, they are all connected together using appropriate fastening elements such as CWELD or CBUSH. In general, quad and hex elements should be preferred over tria, penta and tetra elements. Important features like fillets, holes, and cutouts should be captured in the model appropriately. If there are fasteners or welds between two parallel surfaces, attempts should be made to create a similar mesh on both surfaces. This will facilitate the placement of weld or rigid elements normal to the surfaces without distorting the shell elements. However, many FEA codes support node independent welds, which is based on the tied contact concept. This allows the FEA user to place weld elements independent of the nodes in the parts to be welded. It's recommended to mesh the complex portion of a part first, then proceed towards the simple or plane areas to ensure a good quality mesh in the model. Correct representation of fasteners, joints and welds are necessary in the model for correct load transfer within the structure. The stiffness and preloading should be defined for these elements as applicable for better accuracy. If load transfer is supposed to take place from one surface in a structure to the other, a contact set should be defined between them. Each FEA code has its own format to input contact parameters. A typical contact definition requires master and slave nodes or elements, coefficient of friction, and an offset (gap) distance between surfaces and contact algorithm.

Apply loads and boundary conditions: No matter how good the FE mesh is, the results are not going to be accurate if the FE model is not constrained appropriately or the load applied is not representative of the intended loading. The mesh size and node location sometimes puts constraints on how and where loads and boundary conditions are applied to the model. It would not hurt to re-mesh the FE model locally at the locations where the loads are applied to make sure the loads and boundary conditions are represented in the best possible way.

If the FE analysis is being done to virtually validate any test done in a lab, then it is a good practice to visit the testing facility and take important measurements on test fixtures and loading devices. These measurements will help apply loads and boundary conditions in the FE model in the same way as the part or test specimen is subjected to at the time of testing. For example, measurements of a loading device would help when placing the loads on certain nodes or elements. Similarly, the fixture dimensions would dictate constraint locations (nodes) and its degrees of freedom.

Assign material properties: Material nonlinearity is defined in the FE model in this very important step. The response of the structure depends on the properties supplied to the FE model. The software manual should be referred to understand the input format of the material data card, as different software codes may have different formats. Typically in the lab, loading versus displacement is measured. This information is then expressed as engineering stress and strain. If the software expects the true stress-strain data then the engineering data needs to be converted into true stress-strain data before feeding them to the FE model. Sufficient stress-strain data points should be included to capture the nonlinearity of the material. Analysts should request material suppliers to provide certified properties for the exact same material which is going to be used to build the parts. The properties for commonly used materials can either be pulled from the supplier's website or it could be available at www.matweb.com.

Specify nonlinear analysis control parameters: The basic controlling parameters for nonlinear static analysis are initial increment, minimum and maximum increment, maximum numbers of iterations, the interval at which results file are to be output and convergence criteria for iterations (acceptable residual load, force, displacement).

Run the analysis: The FE model is now ready to be run. The analysis run command may have options to specify the solver version, memory size, and number of CPUs to better control execution.

Review and interpret results: It is highly recommended that the analysis results should be carefully reviewed and checked for accuracy before making any conclusions based on the simulations. There are many ways the FEA results can be checked, some of them are:

- Observe for unexpected movements in the animation.
- Compare the reaction forces against applied forces.
- Check if stresses and strains are as per material properties supplied to the FE model.
- Check interacting surfaces in the contact set for any malfunction.
- Make quick hand calculations by simplifying the problem and compare it with the FE results.

While reporting the FE results, the analysts should always disclose all the assumptions made while building the FE model. In the industrial environment, there is always pressure to complete the FEA sooner to meet tight launch schedules. Still the FEA analysts must check the model thoroughly and review the results carefully before releasing the results to the clients. The FEA results should always be verified using engineering judgment and past results with similar FE models. Unrealistically high or low results should trigger further reviewing of the model. It is highly recommended to maintain consistency in the model to increase accuracy of the results, especially when the analysis is being performed to evaluate the effect of changing certain parameters in the model.

19.7 Nonlinear Analysis With OptiStruct & RADIOSS - Overview

General Remarks

With the 12.0.210 release (October 2013), a renaming of our structural solver products has taken place in order to accommodate the growing trend of applying optimization technologies more intensively in the product development process, instead of just single run analysis. Moving forward, our structural and thermal analysis and optimization product for linear and nonlinear events will be called **OptiStruct** and will continue to leverage the widely used bulk data input format. OptiStruct includes analysis disciplines such as:

- Linear static analysis
- Nonlinear implicit quasi-static analysis
- Linear buckling analysis
- Normal modes analysis
- Complex eigenvalue analysis
- Frequency response analysis
- Random response analysis
- Linear transient response analysis
- Geometric nonlinear implicit analysis
- Linear fluid-structure coupled (acoustic) analysis
- Linear steady-state heat transfer analysis
- Coupled thermal-structural analysis
- Nonlinear steady-state heat transfer analysis

- Linear transient heat transfer analysis
- Contact-based thermal analysis
- Inertia relief analysis with static, nonlinear contact, modal frequency response, and modal transient response analyses
- Component Mode Synthesis (CMS) for the generation of flexible bodies for multi-body dynamics analysis
- Reduced matrix generation
- One-step (inverse) sheet metal stamping analysis
- Fatigue analysis

A typical set of finite elements including shell, solid, bar, scalar, and rigid elements as well as loads and materials are available for modeling complex events.

- Multi-body dynamics solutions integrated via OptiStruct for rigid and flexible bodies include:
- Kinematics analysis
- Dynamics analysis
- Static and quasi-static analysis
- Linearization

All typical types of constraints like joints, gears, couplers, user-defined constraints, and high-pair joints can be defined. High pair joints include point-to-curve, point-to-surface, curve-to-curve, curve-to-surface, and surface-to-surface constraints. They can connect rigid bodies, flexible bodies, or rigid and flexible bodies. For this multi-body dynamics solution, the power of Altair MotionSolve has been integrated with OptiStruct.

Structural Design and Optimization

Structural design tools include topology, topography, and free-size optimization. Sizing, shape and free-shape optimization are available for structural optimization.

In the formulation of design and optimization problems, the following responses can be applied as the objective or as constraints: compliance, frequency, volume, mass, moment of inertia, center of gravity, displacement, velocity, acceleration, buckling factor, stress, strain, composite failure, force, synthetic response, and external (user-defined) functions. Static, inertia relief, nonlinear quasi-static (contact), normal modes, buckling, and frequency response solutions can be included in a multi-disciplinary optimization setup.

Topology, topography, size, and shape optimization can be combined in a general problem formulation.

For more details about OptiStruct optimization capabilities please view the HyperWorks Help Documentation and the free Study Guide - Practical Aspects of Structural Optimization (available on the Academic Website <http://www.altairuniversity.com/academic-program>)

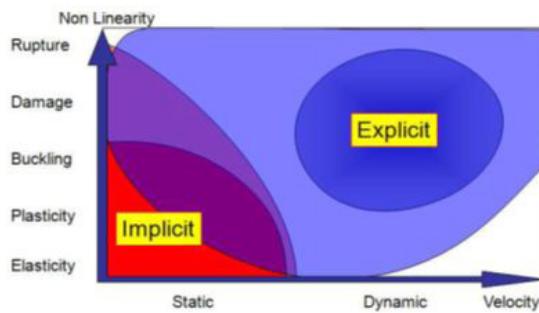
Altair® **RADIOSS®** is a leading structural analysis solver for highly nonlinear problems under dynamic loadings. It is highly differentiated for Scalability, Quality and Robustness, and consists of features for multi-physics simulation and advanced materials such as composites. RADIOSS is used across all industries worldwide to improve the crashworthiness, safety, and manufacturability of structural designs. For over 20 years, RADIOSS has established itself as a leader and an industry standard for automotive crash and impact analysis. RADIOSS will continue to use the well suited block format.

Finite element solutions via Altair RADIOSS include:

- Explicit dynamic analysis
- Nonlinear implicit static analysis

- Transient heat transfer and thermo-mechanical coupling
- Explicit Arbitrary Euler-Lagrangian (ALE) formulation
- Explicit Computational Fluid Dynamics (CFD)
- Smooth Particle Hydrodynamics (SPH)
- Incremental sheet metal stamping analysis with mesh adaptivity
- Linear static analysis
- Normal modes analysis
- Linear and nonlinear buckling analysis

Although there are a range of problems which can be solved with either implicit or explicit solutions, generally, the higher the velocity of the model components, the more suitably it can be modeled with explicit equations. The same is true for the amount of material and geometric nonlinearity. The figure below generally illustrates this concept.



A comparison of explicit and implicit solution domains

Quite obviously, nonlinear effects related to the following do not have to occur concurrently.

- Material (i.e. plasticity)
- Geometry (i.e. large displacements, large strain, large rotation)
- Contact (i.e. changing boundary conditions). For instance, deformations and rotations may be small (infinitesimal), but nonlinearities may be present due to contact or material properties.

Naturally, the user must always understand the dominating nonlinear mechanism in the structure of interest.

19.8 Nonlinear Solution Sequences in OptiStruct

Nonlinear Quasi Static (small deformation)

This solution sequence uses small deformation theory, similar to the way it is used with Linear Static Analysis. Small deformation theory means that strains should be within linear elasticity range (some 5 percent strain), and rotations within small rotation range (some 5 degrees rotation). This also means that there is no update of gap/contact element locations or orientation due to the deformations – they remain the same throughout the nonlinear computations. The orientation may change, however, due to geometry changes in optimization runs. Inertia relief is also possible.

Presently, the sources of nonlinearity include CONTACT interfaces, GAP elements, and MATS1 elastic-plastic material.

Nonlinear Quasi Static (large deformation)

Large displacement nonlinear static analysis is used for the solution of problems wherein the load-response relationship is nonlinear and structural large displacements are involved. The source of this nonlinearity can be attributed to multiple system properties, for example, materials, geometry, nonlinear loading and constraint. Currently, in OptiStruct the following large displacement nonlinear

capabilities are available, including large strain elasto-plasticity, hyperelasticity of polynomial form, contact with small tangential motion, and rigid body constraints.

19.9 Types Of Nonlinearity

Geometric Nonlinearity

In analyses involving geometric nonlinearity, changes in geometry as the structure deforms are considered in formulating the constitutive and equilibrium equations. Many engineering applications require the use of large deformation analysis based on geometric nonlinearity. Applications such as metal forming, tire analysis, and medical device analysis. Small deformation analysis based on geometric nonlinearity is required for some applications, like analysis involving cables, arches and shells. Such applications involve small deformation, except finite displacement or rotation.

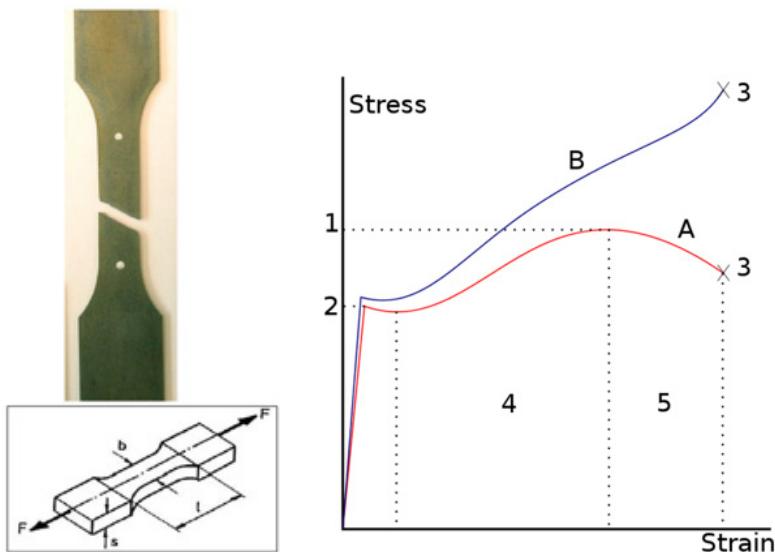
Material Nonlinearity

Material nonlinearity involves the nonlinear behavior of a material based on current deformation, deformation history, rate of deformation, temperature, pressure, and so on.

The MATS1 card is used within a OptiStruct deck to extend the functionality of MAT1 cards to allow for plasticity effects. The MATS1 card is associated with the proper MAT1 card by using the same MID for the MAT1 and MATS1 corresponding entries (We discuss this process in some detail in the paragraph - Modeling Elastic-Plastic Material In OptiStruct 13.0).

What You Need To Know About Material Data* And Their Usage In Simulation - General Recommendations

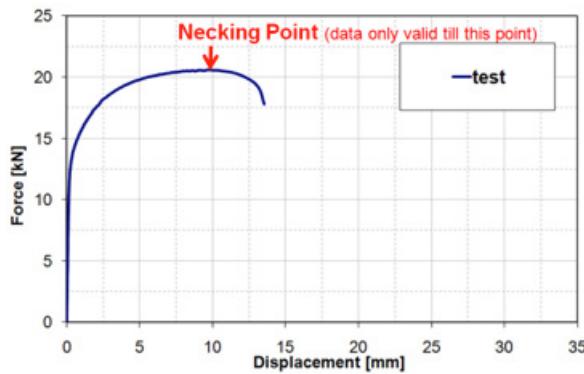
The material properties are characterized and defined in terms of their stress vs. strain relationship.



Left: Ultimate Strength (R_m), Right: Yield Strength (elastic limit), 3: Rupture, 4: Strain Hardening region, 5: Necking region, A: Engineering stress (i.e. constant geometry), B: True stress (i.e. takes into account that cross section of specimen is changing through loading into the plastic region (the image on the right is taken from http://en.wikipedia.org/wiki/File:Stress_v_strain_A36_2.svg; last visit May 28, 2015)

You may find the following publication helpful: STRESS-STRAIN CURVES by David Roylance, Department of Materials Science and Engineering, Massachusetts Institute of Technology <http://web.mit.edu/course/3/3.11/www/modules/ss.pdf> (last viewed April 30, 2015)

Tension Test: SSAB Material: DOCOL 600 DP



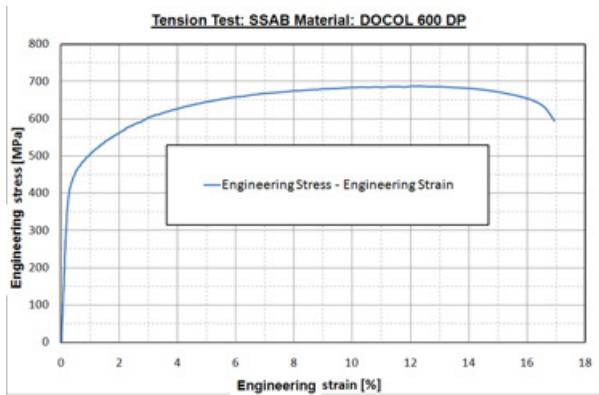
The material test in the lab typically provides forces vs. displacement information

This information needs to be converted into Engineering Stress and Engineering Strain data (geometry is constant). It is quite apparent that the geometry of the specimen needs to be standardized.

Engineering stress and strain are defined as:

$$\sigma_e = F/A_0 \quad \varepsilon_e = \Delta l/l_0$$

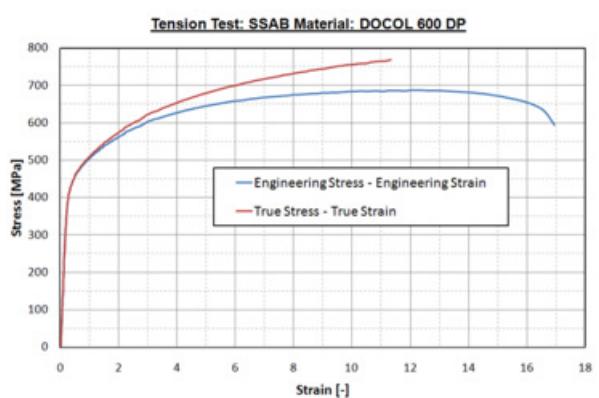
where A_0 is the section of the specimen, l_0 is the initial length, and Δl is the length change.



In the next step, the Engineering Stress and Strain is translated into True Stress and True Strain.

$$\sigma_{tr} = \sigma_{tr} \exp(\varepsilon_{tr}) \quad \varepsilon_{tr} = \ln(1 + \varepsilon_e)$$

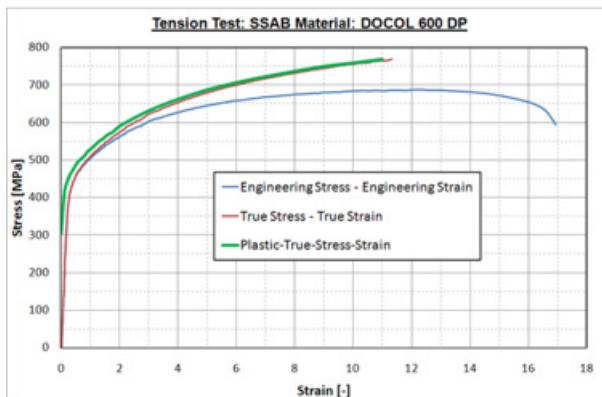
$$\sigma_{tr} = \sigma_e (1 + \varepsilon_e)$$



And eventually, the True Stress – True Strain must be expressed as True Stress vs. True Plastic Strain (i.e. subtract the elastic part of the curve):

$$\varepsilon_{pl} = \varepsilon_{tr} - \sigma_{tr}/E$$

Note: E (Youngs modulus) is determined from the curve e.g. $E = \text{yield stress } R_e / \varepsilon_e$



Constraint and Contact Nonlinearity

Constraint nonlinearity in a system can occur if kinematic constraints are present in the model. The kinematic degrees-of-freedom of a model can be constrained by imposing restrictions on its movement. In OptiStruct, constraints are enforced with Lagrange multipliers.

In the case of contact, the constraint condition is based on inequalities and such a constraint generally does not allow penetration between any two bodies in contact.

A summary on Contact Simulation provided further below in the Example - Contact Analysis in OptiStruct 13.0.

General Remarks

Nonlinear problems are generally history dependent. In order to achieve a certain level of accuracy, the solution must be obtained in a series of small increments. For this purpose we need to solve the equilibrium equation at each increment and a corresponding increment size is selected.

Newton's method is used to solve the nonlinear equilibrium equation in OptiStruct. If the solution is smooth, quadratic of rate of convergence may be achieved when compared with other methods. This method is also very robust in highly nonlinear situations.

Choosing a suitable time increment is very important. In OptiStruct, an automatic time increment control is available. It should be suitable for a wide range of nonlinear problems and, in general, is a very reliable approach.

The automatic time increment control functionality measures the difficulty of convergence at the current increment. If the calculated number of iterations is equal to the optimal number of iterations for convergence, OptiStruct will proceed with the same increment size. If a lesser number of iterations is required to achieve convergence, the increment size will be increased for the next increment. Similarly, if it is determined that too many iterations are required, the current increment will be attempted again with a smaller increment size.

Large displacement nonlinear analysis is supported only for solid elements, RROD, RBAR, RBE2, and RBE3 entries

Large Displacement Nonlinear Analysis is not supported in conjunction with the following elements:

(a) The following elements can exist in the model, but they will be resolved using small-displacement nonlinear theory (equivalent to ANALYSIS = NLSTAT with plasticity). If large translations or rotations are expected on these elements, then the results may not be accurate: Shell, Gasket, and Bushing elements

(b) The following elements are not allowed and OptiStruct will error out if they are present: CBAR, CBEAM, CGAP, CGAPG, CWELD, CSEAM, CFAST, RBE1, CROD, CELAS, and CONM

19.10 Modeling Elastic-Plastic Material In OptiStruct 13.0

(Modified version; original authors are Kristian Holm and Christian Steenbock, Altair Germany)

Non-Linear Elastic Plastic Material Definition In OptiStruct

The MATS1 card specifies stress-dependent and temperature-dependent material properties for use in applications involving nonlinear materials.

It enhances the linear elastic material MAT1 by plasticity effects, i.e. a MAT1 Material definition is needed. The MATS1 Card is then a kind of extension within the MAT1 definition.

What you need to know about MATS1 (with regard to OptiStruct 13):

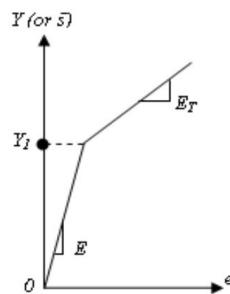
- MATS1 is not available for 1D
- MATS1 can be used for 2D and 3D elements provided we consider small displacements
- MATS1 is supported for 3D elements and large displacements
- MATS1 is not supported for second order shell elements
- Kinematic hardening and Mixed hardening are supported only for solids

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
MATS1	MID	TID	TYPE	H	YF	HR	LIMIT1		
	TYPSTRN								

The screenshot shows the OptiStruct software interface. The Entity browser on the left lists various project components like Assembly Hierarchy, Card (2), Component (2), Load Collector (13), Load Step (11), Material (1), Property (1), and Title (1). The Material (1) node is expanded, showing a sub-node 'elastic_plastic' with ID 1. The main window displays a table of material properties:

Name	Value
Solver Keyword	MAT1
Name	elastic_plastic
ID	1
Color	[Master Model]
Include File	MAT1
Card Image	Do Not Export
User Comments	210000.0
E	
G	
NU	0.3
RHO	
A	
TREF	
GE	
ST	
SC	
SS	
MATS1	
TID	
TYPE	PLASTIC
H	
YF	
HR_REAL	
HR	
LIMIT1	
TYPSTRN	1

For elastoplastic materials, the elastic stress-strain matrix is computed from a MAT1 entry, and the isotropic plasticity theory is used to perform the plastic analysis. In this case, either the table identification TID (field 3 above) or the work hardening slope H (field 5 above) may be specified, but not both. If the TID is omitted, the work hardening slope H must be specified, unless the material is perfectly plastic. The plasticity modulus (H) is related to the tangential modulus (ET) by



A simply modeled plasticity region defined by E_T and Y_1 .

E_T is the slope of the uniaxial stress-strain curve in the plastic region as a continuation of E past the yield point. Using E_T and E , it is possible to calculate H (i.e. plasticity hardening) by the equation:

$$H = \frac{E_T}{1 - \frac{E_T}{E}}$$

Hence, there are two ways of defining the elasto-plastic material behavior:

- One parameter model with a constant slope in the plastic region (H defined; field 5). With H - Work hardening slope (slope of stress versus plastic strain) in units of stress. For elastic-perfectly plastic cases, $H = 0.0$ (default).

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
MATS1	MID	TID	TYPE	H	YF	HR	LIMIT1		
	TYPSTRN								

b. For more than a single slope in the plastic range, the stress-strain data must be supplied on a TABLES1 entry referenced by TID (field 3).

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
MATS1	MID	TID	TYPE	H	YF	HR	LIMIT1		
	TYPSTRN								

If TID (ID of TABLES1 card) is given, TABLES1 entries (ε_x, Y_x) must be defined in the first quadrant. The data points must be in ascending order. The TABLES1 is defined as a load connector with Card Image TABLES1.

What is the format of the data in TABLES1, what kind of data is needed?

This is defined by the TYPSTRN parameter:

TYPSTRN = 0 = total strain

- First point must be at the origin ($x_1=0, y_1=0$)
- Second point must be at the initial yield point ($y_2=LIMIT1$)
- The slope of the line joining the origin to the yield stress must be equal to the value of E

TYPSTRN = 1 = plastic strain

- First point at $x_1=0$ must have $y_1=LIMIT1$

The LIMIT1 field (field 8; yield stress) can be blank if the initial yield point value is defined via a referenced TABLES1 entry on the TID field

The stress – strain curve of the plastic region defined by discrete data values in TABLES1

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
TABLES1	TID								
	x1	y1	x2	y2	x3	y3	x4	y4	
	x5	y5	...						

TID: ID of TABLES1 card

In the tutorial about the tensile test of elasto-plastic test sample we will show how to create such a TABLES1 based on for instance, data provided in an EXCEL sheet automatically.

Note: For analysis where small deformations are assumed, there should be little or no difference between the true stress-strain curve and the engineering stress-strain curve, so either of them may be used in the TABLES1 definition. For analyses where small deformations are not assumed, the true stress-strain curve should be used.

If the deformations go past the values defined in the table, the curve is extrapolated linearly.

General parameters:

MID (field 2): ID of respective MAT1

TYPE (field 4): PLASTIC/NLELAST (Currently only PLASTIC available for NLSTAT)

YF (field 6): Yield function criterion = von Mises (only option right now)

HR (field 7): Hardening rule

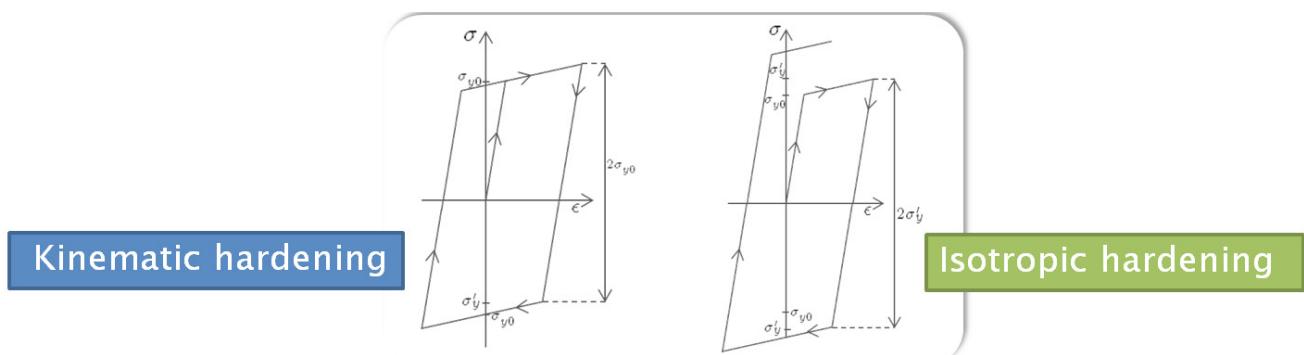
1 = isotropic hardening

2 = kinematic hardening

3 = mixed hardening (30% kinematic and 70% isotropic hardening)

0<HR<1 = mixed hardening (HR*100% kinematic and (1-HR)*100% isotropic hardening)

Note: Kinematic/Mixed hardening only available for solid elements right now



Aside the definition of the elastic-plastic material characteristics, some additional solver (OptiStruct) parameters are needed.

NLPARM = Card image of a load collector; defines parameters for Nonlinear Static Analysis (or Heat Transfer Analysis). The NLPARM card (=load collector) will be referenced in the loadstep/subcase definition.

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)
NLPARM	ID	NINC			KSTEP	MAXITER	CONV	
	EPSU	EPSP	EPSW			MAXLS		LSTOL

NINC

NINC	Number of implicit load sub-increments Default = 1 (no increments) for ANALYSIS = NLSTAT and ANALYSIS=NLHEAT Default = 10 for ANALYSIS = NLGEOM (Integer > 0)
------	--

The solution method for quasi-static nonlinear analysis (ANALYSIS = NLSTAT, defined in the respective loadstep) is full Newton. The stiffness matrix is updated at each iteration. NINC > 0 represents the number of equal subdivisions that the total load in a given subcase will be divided into. If NINC is blank, the entire load for a given subcase is applied at once. The Newton method will be applied to consecutive load levels until the final load is reached.

MAXITER

MAXITER	Limit on number of implicit iterations for each load increment. If reached, the solution is terminated (ANALYSIS = NLSTAT and ANALYSIS=NLHEAT). Default = 25 (Integer > 0)
---------	---

CONV

CONV	<p>Flags to select implicit convergence criteria.</p> <p>Default = UPW for ANALYSIS = NLSTAT</p> <p>Default = PW for ANALYSIS = NLGEOM</p> <p>(Any combination of U, P and W)</p>
------	--

U: Displacement P: Load W: Work

EPSU, EPSP, EPSW

Error tolerances

MAXLS

Maximum number of line search iterations

LSTOL

Line search tolerance

Loadstep Definition

In the image below the set-up of a loadstep/subcase is depicted.

We look at the loadstep named „nl_5“ with ID=6. The loadcollector „nlparm (7)“ which bears the NLPARM definition (from above) is now referenced by the NLPARM parameter. The ANALYSIS TYPE is set to Non-linear Quasi Static / NLSTAT

The screenshot shows the ANSYS Workbench interface with the Entities browser open. The browser tree includes Assembly Hierarchy, Card (2), Component (2), Load Collector (13), and Load Step (11). Under Load Step (11), there are seven entries: nl_0, nl_1, nl_2, nl_3, nl_4, nl_5, and nl_6. The nl_5 entry is selected and highlighted in blue.

Below the browser, a detailed view of the nl_5 loadstep parameters is shown in a table:

Name	Value
Solver Keyword	SUBCASE
Name	nl_5
ID	6
Include File	[Master Model]
User Comments	Do Not Export
Subcase Definition	
Analysis type	Non-linear quasi-static
SPC	spc_085 (12)
LOAD	<Unspecified>
SUPPORT	<Unspecified>
DEFORM	<Unspecified>
PRETENSION	<Unspecified>
MPC	<Unspecified>
TEMP	<Unspecified>
NLPARM	nlparm (7)
STATSUB (PRETENS)	<Unspecified>
SUBCASE OPTIONS	
LABEL	<input checked="" type="checkbox"/>
label	loadstep1
SUBTITLE	<input type="checkbox"/>
ANALYSIS	<input checked="" type="checkbox"/>
TYPE	NLSTAT
CNTNLNSUB	<input checked="" type="checkbox"/>
OPTION	YES
EIGVRETRIEVE	<input type="checkbox"/>
EIGVSAVE	<input type="checkbox"/>
EXCLUDE	<input type="checkbox"/>
RADSND	<input type="checkbox"/>
RESVEC	<input type="checkbox"/>
SOLVTYP	<input type="checkbox"/>
OUTPUT	<input checked="" type="checkbox"/>
ACCELERATION	<input type="checkbox"/>
CSTRAIN	<input type="checkbox"/>
CSTRESS	<input type="checkbox"/>
DAMAGE	<input type="checkbox"/>
DISPLACEMENT	<input type="checkbox"/>
EDE	<input type="checkbox"/>
EKE	<input type="checkbox"/>

Note: In the OUTPUT panel (to be activated within the load step panel) additional output data/information such as SPC Forces can be requested. Results are written per individual load step – not per increment or per iteration! In other words, if you like to see not only the results of the final iteration, the load must be split-up into several loadsteps!

Optionally, the following parameter may be defined in the load step

CNTNLSUB

Depending on whether a more complex loading path needs to be considered, e.g. in case of path-dependent problems (as with plasticity or contact with friction), a subcase continuation card CNTNLSUB can be activated within the loadstep.

CNTNLSUB, YES:

This nonlinear subcase continues the solution from the nonlinear subcase preceding (it must be the same analysis type). "Preceding" refers to the sequence in the input deck and NOT the numbering of the sub cases

CNTNLSUB, NO:

This nonlinear subcase executes a new solution sequence starting from the initial, stress-free state of the model.

CNTNLSUB, SID:

This nonlinear subcase continues the nonlinear solution from the subcase with the ID given through SID. The subcase must precede the current subcase in the deck and must be a nonlinear subcase of the same type.

```

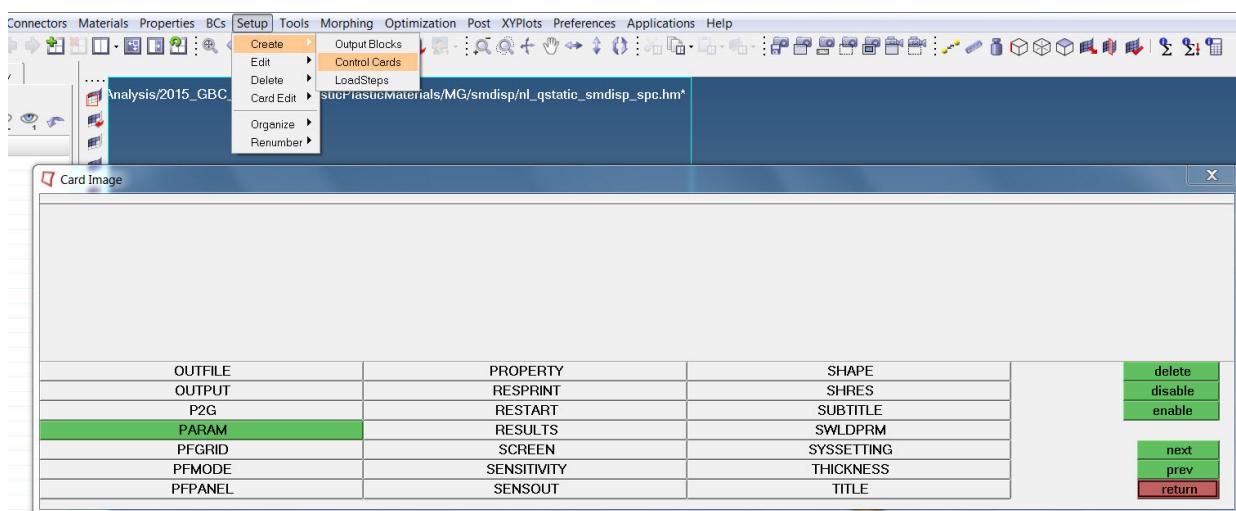
36 CNTNLSUB YES
37 SPC = 1
38 LOAD = 20
39 NLPARM = 3
40 $
41 $HNAME LOADSTEP 3"loadstep3" 9
42 $
43 SUBCASE 3
44 LABEL loadstep3
45 ANALYSIS NLSTAT
46 CNTNLSUB YES
47 SPC = 1
48 LOAD = 30
49 NLPARM = 3
50 $
51 $HNAME LOADSTEP 4"loadstep4" 9
52 $
53 SUBCASE 4
54 LABEL loadstep4
55 ANALYSIS NLSTAT
56 CNTNLSUB YES
57 SPC = 1
58 LOAD = 40
59 NLPARM = 3
60 $
61 $HNAME LOADSTEP 5"loadstep5" 9
62 $
63 SUBCASE 5
64 LABEL loadstep5
65 ANALYSIS NLSTAT
66 CNTNLSUB YES
67 SPC = 1
68 LOAD = 50
69 NLPARM = 3
70 $
71 $HNAME LOADSTEP 6"loadstep6" 9
72 $
73 SUBCASE 6
74 LABEL loadstep6
75 ANALYSIS NLSTAT
76 CNTNLSUB YES
77 SPC = 1

```

In addition the option **Expert System** may be activated

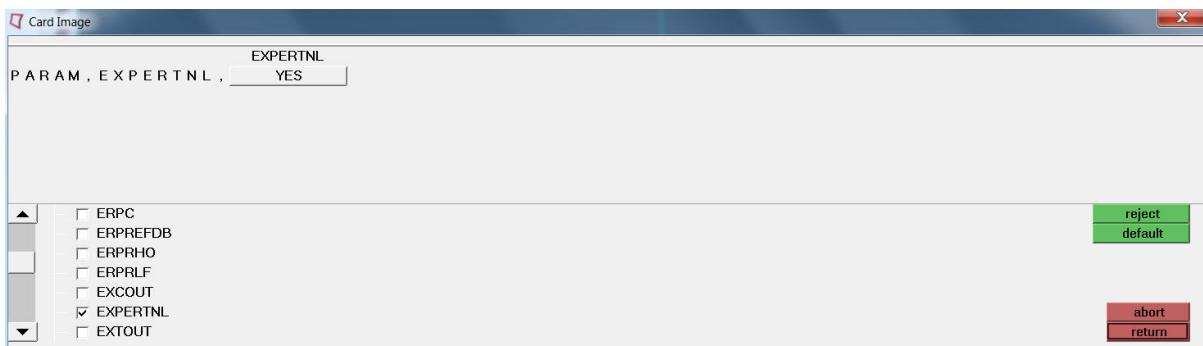
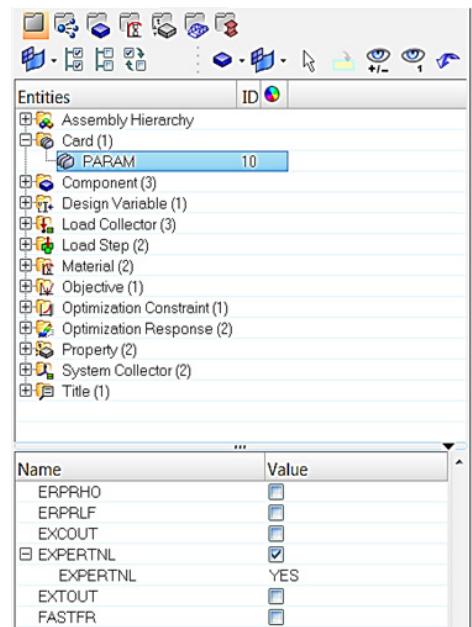
PARAM, EXPERTNL, YES (

Activates an „expert system“ that aids in the convergence of small displacement nonlinear problems (NLSTAT)



Detailed information on EXPERTNL is provided in the image:

Parameter	Values	Description
EXPERTNL	YES, NO, CNTSTB, AUTO Default = AUTO	<p>YES activates a nonlinear "expert system" that aids in the convergence of small displacement nonlinear problems (NLSTAT). This version is designed to facilitate obtaining converged, high accuracy solutions without much concern for computational time. The system monitors the convergence of nonlinear processes and, if needed, implements measures designed to improve convergence for poorly converging cases. These measures include: performing additional iterations, under-relaxation, automatic adjustment of the load increment, as well as backing off to the last convergent solution and retrying. This may lead to a large number of nonlinear iterations for poorly converging problems.</p> <p>CNTSTB additionally introduces temporary stabilization on contact interfaces (CONTACT or GAP(G) elements) that may improve nonlinear convergence, especially in cases where individual parts lack full support and are supported only by contact. The stabilization is applied only during incremental loading and is not present in the final solution for the respective nonlinear subcase.</p> <p>AUTO activates a "light" version of the expert system, which is designed to facilitate converging nonlinear process in reasonably close to minimum number of iterations. In particular, this version may adjust the time step, including increasing the time step beyond that prescribed on the NLPARM card.</p> <p>This parameter does not apply to geometric nonlinear solution sequences (NLGEOM, IMPDYN or EXPDYN).</p>

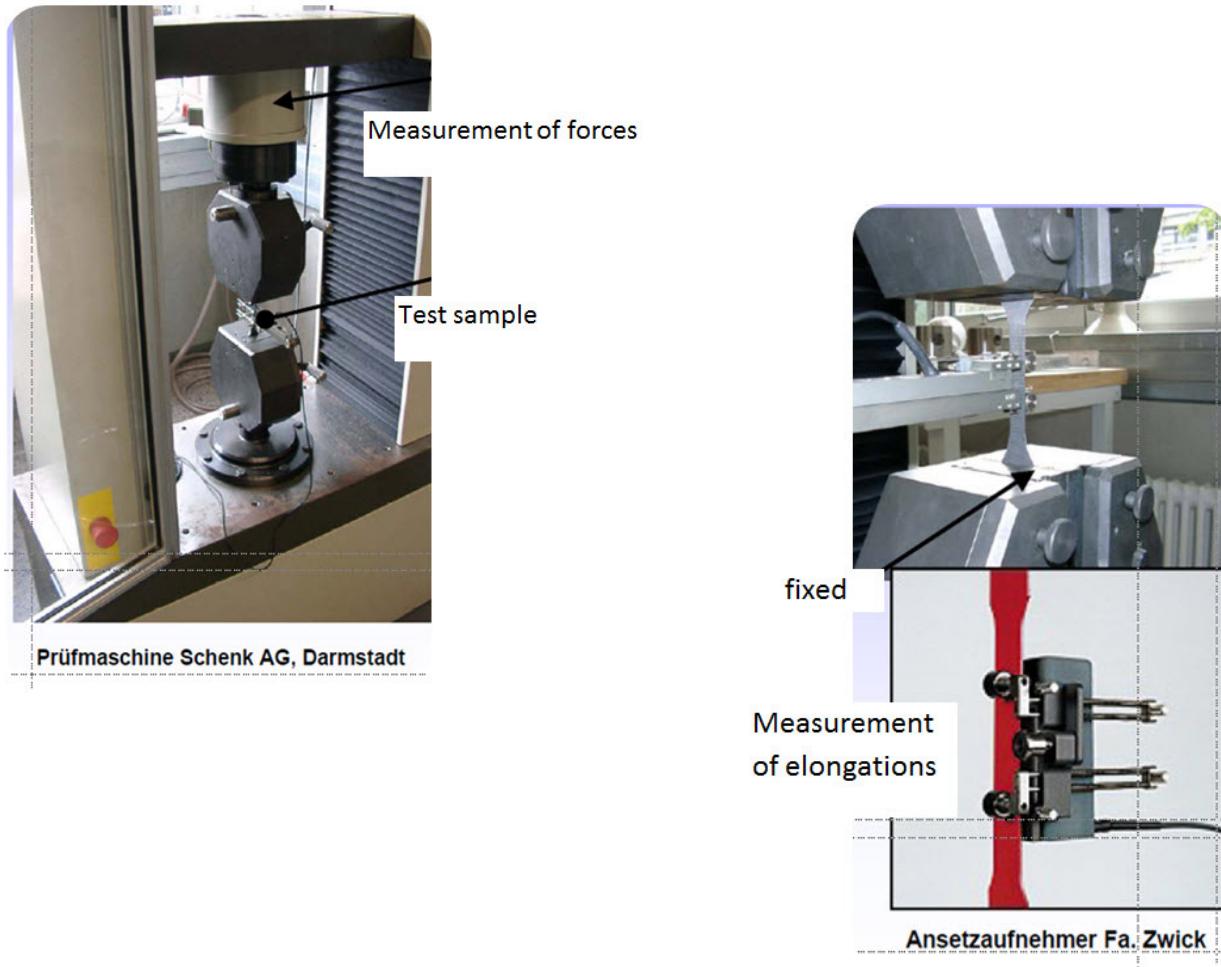


19.11 EXAMPLE: TENSILE TEST (elastic-plastic)

Before calling up HyperWorks and importing model files, please take a second and think about:

- How is the tensile test typically conducted?
- How is the test sample loaded?
- Where and how do you measure the relevant data, e.g. the extension of the sample?
- Do you look at true stress/strain or engineering stress/stain?
- What are likely mistakes/errors while conducting the test?
- Do measure a set of samples?
- Which simulation results (i.e. type of information) are needed?

The test set-up in the lab is as shown:



Available

The model is readily meshed with hexahedral elements, an RBE2 element (shown in green) for application of enforced displacements is also included (more information about RBE2 elements are included in this study guide).

Model file: tensile_smdisp_start.hm



The material data (stress-strain curve; file: stress_strain.csv) as well as the test data (file: force vs. displacement; test_results.csv) are given for your reference (the files will be uploaded to the Academic Training Center).

What you need to do:

- Define elastic-plastic material (MATS1 with TABLE1)
- Apply boundary constraints and enforced displacements
- Define NLPARM load collector
- Create loadsteps (NLSTAT)
- Requests output (stress, strain, SPCForces)
- Postprocess results (in HyperView and HyperGraph)

a. Definition of elastic-plastic material properties

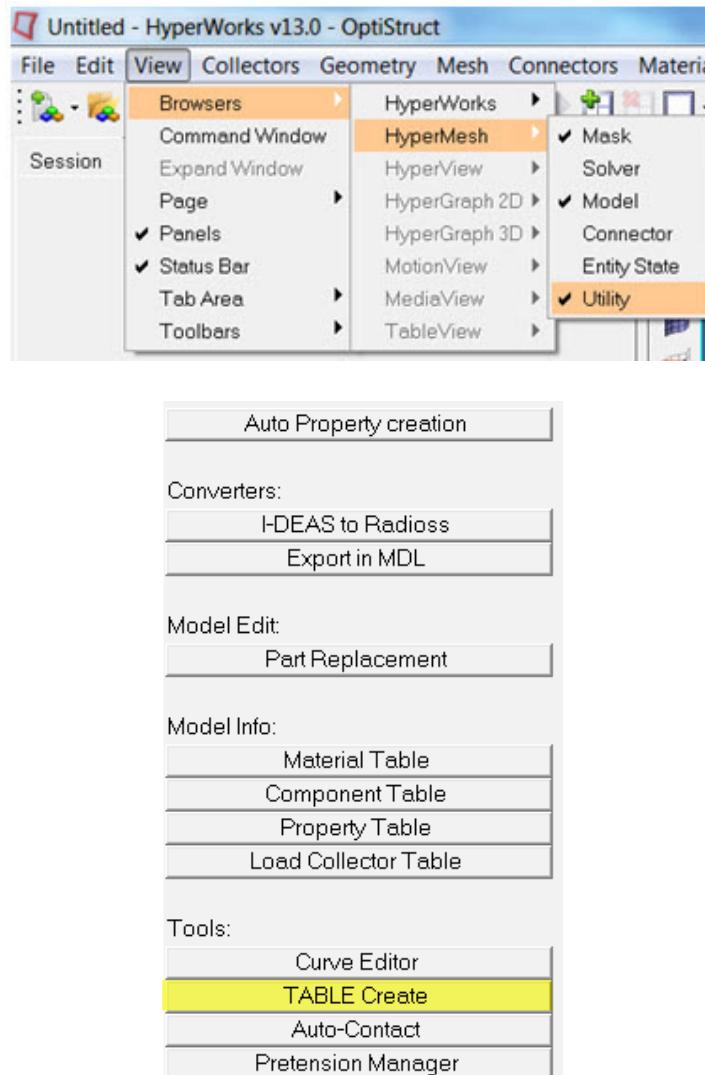
The stress-strain data are given in the following format (comma separated; Excel sheet)

	A	B
1	0.0000,306.0	
2	0.0011,415.0	
3	0.0022,445.0	
4	0.0030,461.0	
5	0.0040,474.0	
6	0.0052,489.0	
7	0.0061,498.0	
8	0.0071,505.0	
9	0.0081,512.0	
10	0.0090,522.0	
11	0.0102,530.0	
12	0.0121,543.0	

As $x_1=0$ and $y_1=306$ it is obvious (from above discussion) that we look at plastic stress-strain. Hence, in the MATS1 Card we have to set the parameter TYPSTRN = 1.

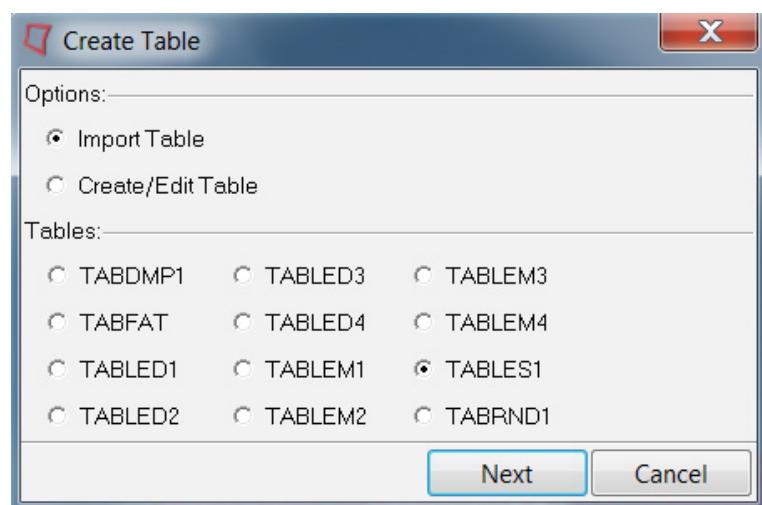
Question: There are 34 data pairs in the Excel sheet. How can we create the TABLES1 most efficiently?

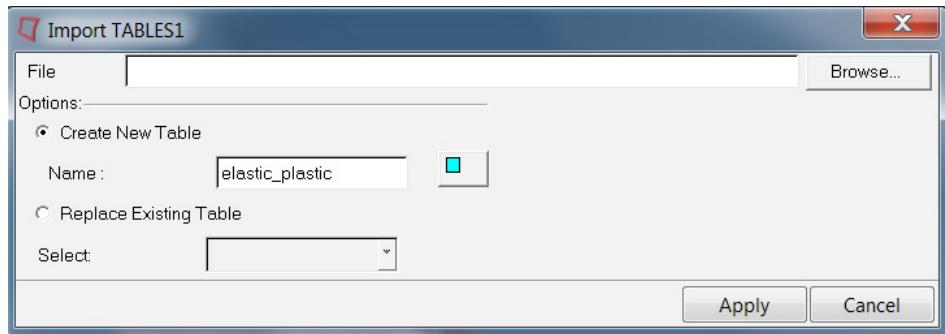
- Manual copy & paste (feasible but not recommended)
- Employ the “TABLE Create” script available under View > Browsers > HyperMesh > Utility



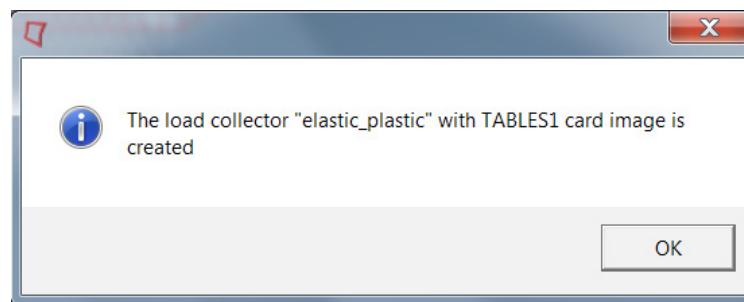
Activating the “TABLE Create” script launches the window as shown.

Select „Import Table“ and TABLES1.





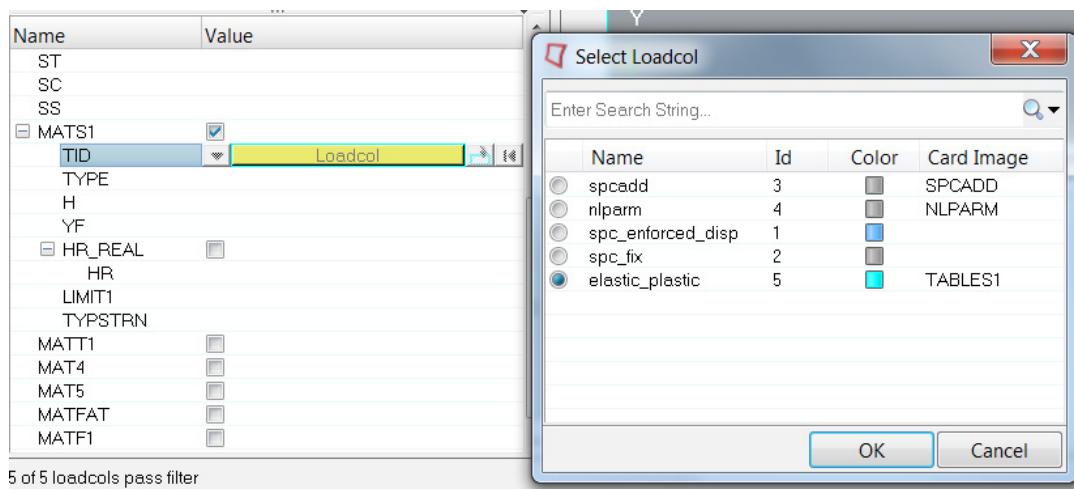
Select the Excel file from your directory, provide a Name of the load collector to be created once you hit the Apply button (here the load collector is named „elastic_plastic“, its Card Image is TABLES1)



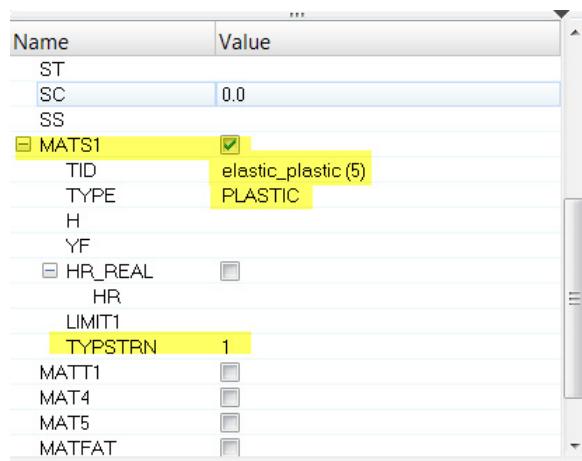
The TABLES1 now lists the following stress strain data (plastic domain)

x	y
1 0.0	306.0
2 0.0011	415.0
3 0.0022	445.0
4 0.003	461.0
5 0.004	474.0
6 0.0052	489.0
7 0.0061	498.0
8 0.0071	505.0
9 0.0081	512.0
10 0.009	522.0
11 0.0102	530.0
12 0.0121	543.0
13 0.013	550.0
14 0.014	555.0
15 0.015	561.0
16 0.0159	567.0
17 0.0171	572.0
18 0.0181	577.0
19 0.0204	592.0
20 0.0303	632.0
21 0.0405	663.0
22 0.0502	687.0
23 0.06	706.0
24 0.0702	722.0
25 0.0807	737.0
26 0.09	749.0
27 0.0997	758.0
28 0.101	759.0
29 0.11	768.0
30 0.15	805.0
31 0.2	840.0
32 0.3	900.0
33 0.5	1000.0
34 1.0	1210.0

The TABLES1 (which is a loadcollector) needs to be referenced in a material collector of Type: MAT1 (see below)

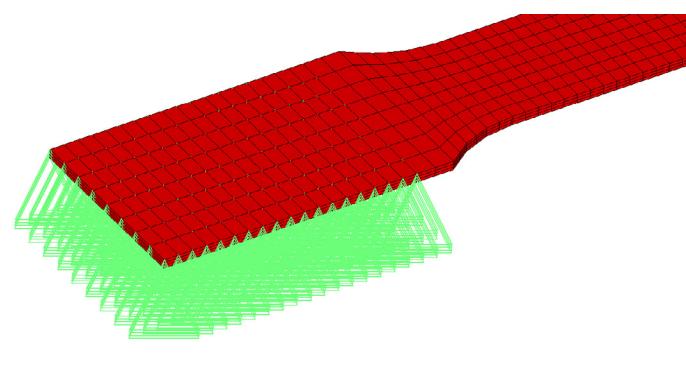


Eventually, you need to define TYPE and TYPSTRN=1 (because we consider Plastic Strain data and not Total Strain data)

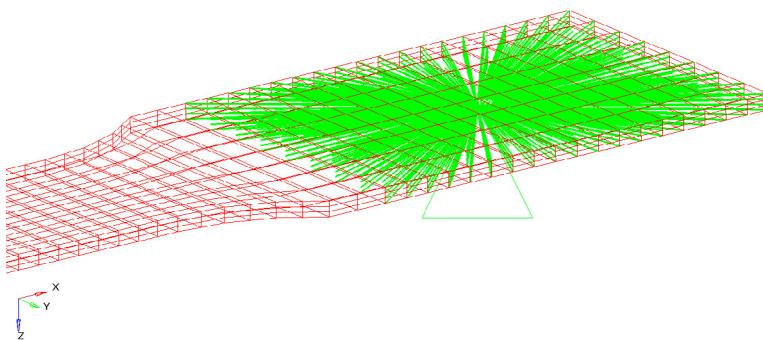


b. Model Constraints & Loadsteps

The sample will be pinned at one end (dof 1-3, translational dof are fixed)



An enforced displacement (total of 17 mm) is applied to the Master Node of the RBE2 element



Question: How to define enforced displacements?



Simply assign the corresponding value to the dof of interest (here the enforced displacement is in x-direction and is set to 1.7 mm). Mind the load type is set to SPC (single point constraint)

Note: The enforced displacement constraint AND the other constraints from before are ALL placed in the SAME load collector!

However, please recall the information about OUTPUT request provided above. Since we are interested in the intermediate results (i.e. results corresponding to an enforced displacement of for instance, 1 mm, 2 mm, 5 mm etc.) we need to define a set of corresponding loadcollectors and loadsteps.

In this particular model, the magnitude of enforced displacements in x-direction is 17 mm. Quite arbitrarily and for simplicity reasons, the enforced displacements have been split up in 10 loadsteps as OptiStruct writes out results per loadstep. This implies that we will have 10 data points at the end (not really sufficient to track the load curve of the test). Of course, this also means that we need to create 10 load collectors (to account for the different enforced displacements). Note: If more intermediate results are needed more loadsteps must be defined.

Loadstep 1: enforced displacement dof1 (x-direction) = 1.7 mm at RBE2; the other end of the sample is spinned dof 1-3 =0

Loadstep 2: enforced displacement dof1 (x-direction) = 3.4 mm at RBE2; the other end of the sample is spinned dof 1-3 =0; plus subcase continuation card CNTNLSUB

Loadstep 2: enforced displacement dof1 (x-direction) = 5.1 mm at RBE2; the other end of the sample is spinned dof 1-3 =0; plus subcase continuation card CNTNLSUB

....

....

Loadstep 10: enforced displacement dof1 (x-direction) = 17 mm at RBE2; the other end of the sample is pinned dof 1-3 =0; plus subcase continuation card CNTNLSUB

(reminder: the load collector "elastic_plastic" contains the TABLES1 information; nlparm is explained further below)

Load Collector (13)	
elastic_plastic	5
nlparm	7
spc_0	2
spc_017	8
spc_034	9
spc_051	10
spc_068	11
spc_085	12
spc_102	13
spc_119	14
spc_136	15
spc_153	16
spc_170	17

In the Subcase Definition the „Analysis type“ is set to „non-linear quasi-static“ which automatically enables the NLSTAT command in the Subcase Options section; SPC references the corresponding loadcollector (with enforced displacement and pinned end of the sample).

The CNTLSUB command is activated in order to continue the nonlinear solution from the preceding nonlinear subcase.

The NLPARM command in the loadstep definition references the loadcollector named „nlparm (7)“ and defines parameters for the nonlinear static analysis (or heat transfer analysis solution).

The screenshot displays the ANSYS Workbench interface with two main windows:

- Entity Browser (Left):** Shows the assembly hierarchy, including Card (2), Component (2), and Load Collector (13). A specific entry under Load Collector is highlighted: "elastic_plastic nlparm 7".
- Property Manager (Right):** Contains several tabs of configuration data:
 - Subcase Definition:** Set to "Non-linear quasi-static" with "spc_017 (8)" as the SPC.
 - SUBCASE OPTIONS:** Includes settings for "LABEL" (loadstep1), "ANALYSIS" (NLSTAT), and "CNTLSUB" (YPS).
 - Load Step (11):** A table showing 10 load steps labeled nl_1 through nl_10.
 - Material (1):** Set to "elastic_plastic" with ID 1.
 - Property (1):** Set to "elastic_plastic" with ID 1.
 - Title (1):** Set to "Model Info" with ID 1.
 - Model Info:** A detailed table of parameters including NINC, KSTEP, MAXITER, CONV, EPSU, EPSP, EPSW, and MAXLS.

With:

NINC = Number of implicit load sub-increments; Default = 1 (no increments) for ANALYSIS = NLSTAT. Increasing NINC may help achieve convergence.

MAXITER = Limit on number of implicit iterations for each load increment. If reached, the solution is terminated (ANALYSIS = NLSTAT and ANALYSIS=NLHEAT); Default = 25

CONV = Flags to select implicit convergence criteria; Default = UPW for ANALYSIS = NLSTAT

EPSU = Error tolerance for displacement (U) criterion; Default = 1.0E-3 for ANALYSIS = NLSTAT

EPSP = Error tolerance for load (P) criterion; Default = 1.0E-3 for ANALYSIS = NLSTAT (and ANALYSIS=NLHEAT)

EPSW = Error tolerance for work (W) criterion; Default = 1.0E-7 for ANALYSIS = NLSTAT (and ANALYSIS=NLHEAT)

MAXLS = Maximum number of line searches allowed for each iteration; Default = 0 for ANALYSIS = NLSTAT

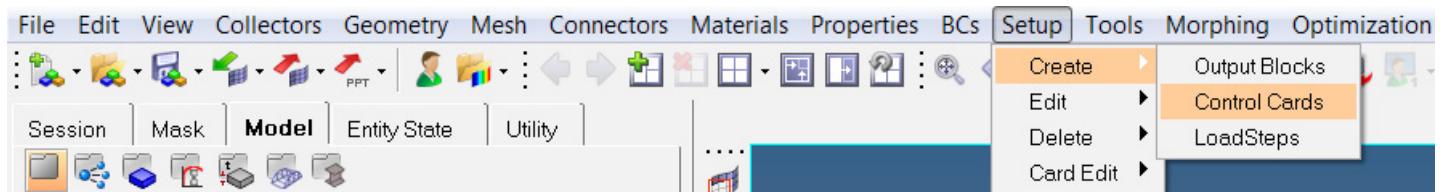
LSTOL = Line search tolerance; Default = 1.0E-3

OUTPUT Control

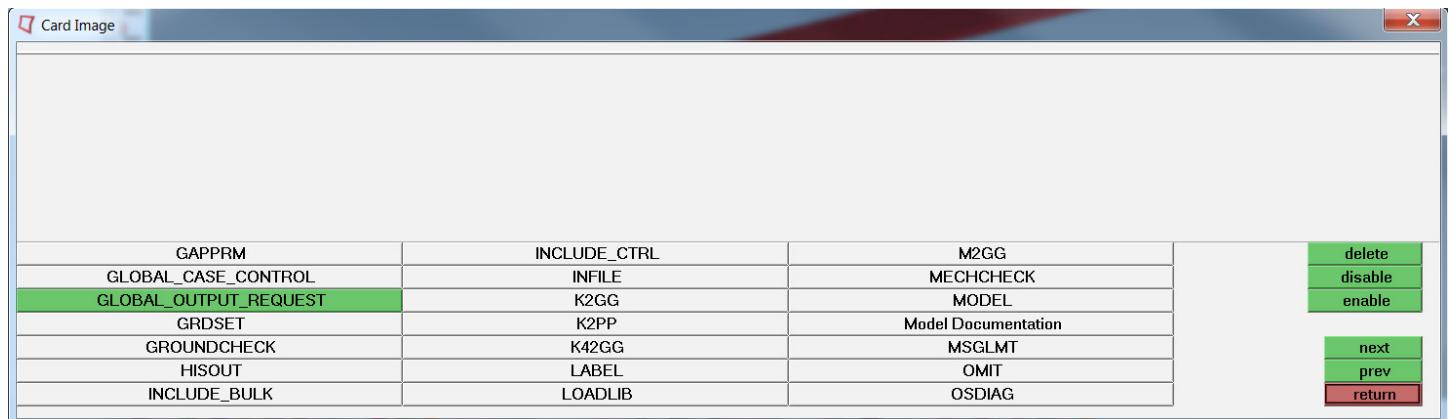
As questioned in the intro to this tutorial already – which results are needed at all?

1. Elongation between two reference points on the sample: this can be measured in HyperView directly, i.e. contouring displacements and then creating a measure in HyperView of type „Distance between“ (see test set-up)
2. What are the forces in the model (recall: test data record force vs. displacement)?

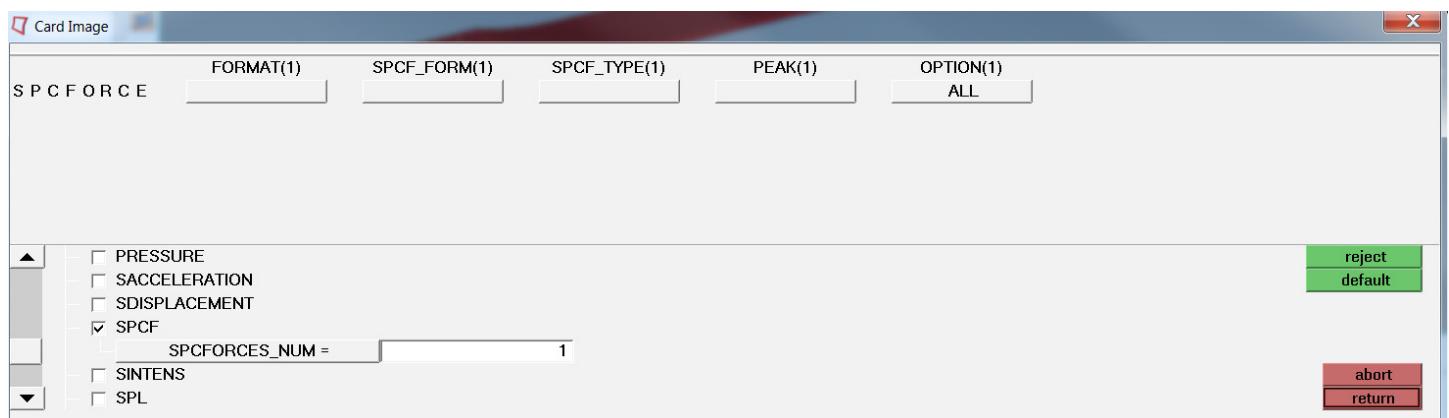
To address 2. we need to activate the SPCFORCE command (default: off) in the Control Cards menu:



Activate GLOBAL_OUTPUT_REQUEST ...



And select SPCFORCE ...



Finally, save the model (*.hm format) and start the analysis.

In the *.out file the analysis history is documented (never miss to view the out file).
 The analysis converged sucessfully, however we learn that:

*** Nonlinear solution has converged ***

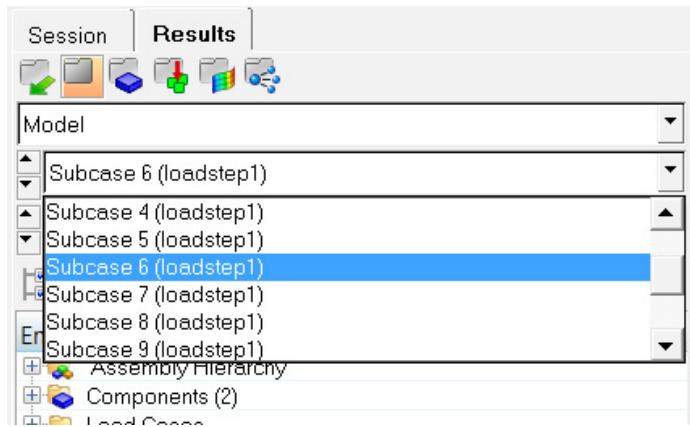
*** WARNING # 3200

Maximum plastic strain in this solution: 0.103 exceeds the limit of small deformation theory. The validity of this solution is questionable.

Nevertheless, let's have a closer look at the results using HyperView and HyperGraph

Postprocessing With HyperView and HyperGraph

Quite obviously we can only display results with respect to a particular loadstep (by default), respectively.



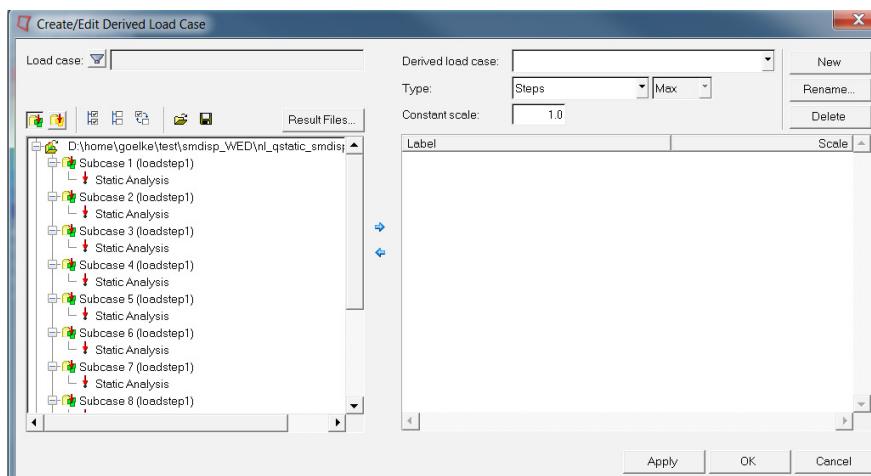
As a consequence, it is not possible to display the complete Force vs. Displacement history of the simulation in a single graph as we would get in a transient analysis. In other words, we can show results of a particular loadstep but we can't create an animation which inlcudes (shows) the entire loading cycle.

To overcome this „limitation“ HyperView offers the option to create a „Derived Loadstep“

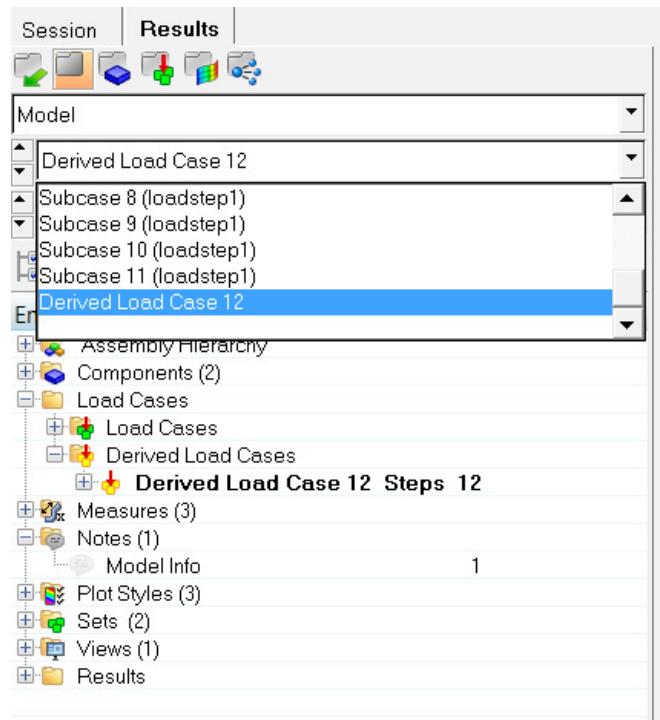


What does it mean?

By creating a derived loadstep the results from a selection of loadsteps (here we select all loadsteps) will be „merged“ into 1 single new (artifical) loadstep. In this way the quasi-static results from the analysis before can be visualized similar to the results of a transient analysis.



Select the loadsteps/subcase of interest (here All) in the left window, then click the blue arrow pointing to the right (as Type select Steps). A new derived loadstep will be created and will be listed as a „regular“ loadstep.

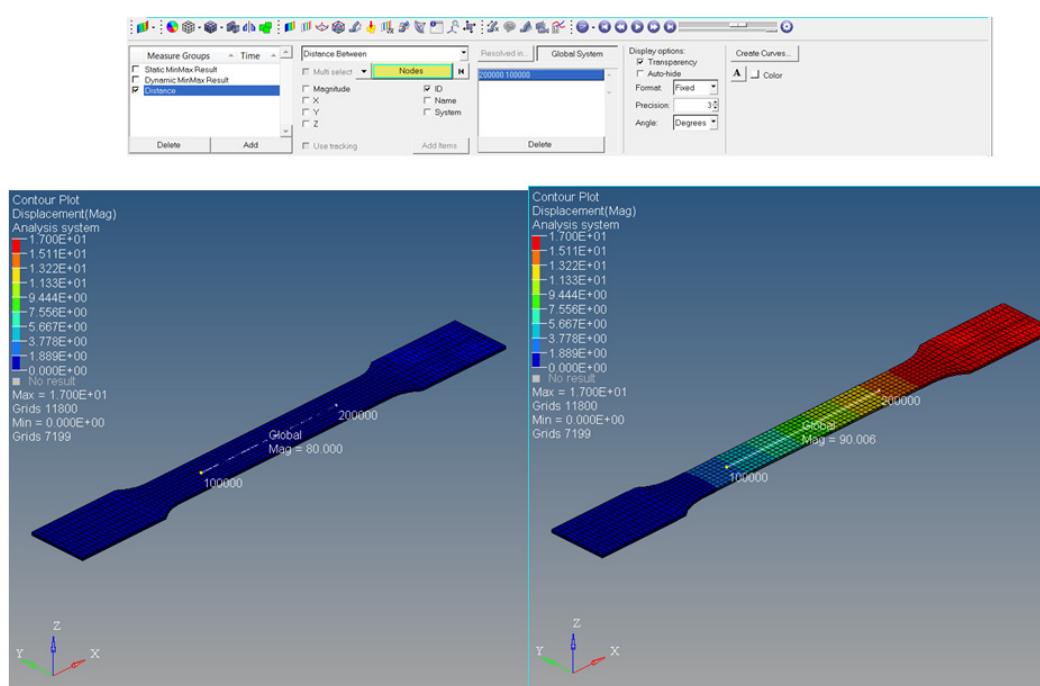


Make sure and export this model (with the derived loadstep) as an additional *.h3d file (needed in HyperGraph)

This loadstep can now be animated in HyperView

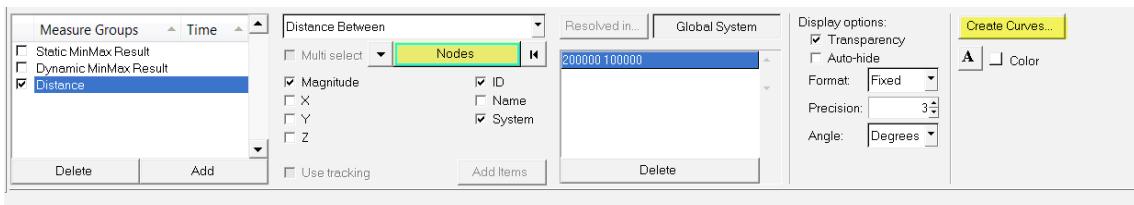


To measure the elongation of the sample due to the enforced displacements, create a measure between the nodes with ID 200000 and ID 100000; initial distance 80 mm.

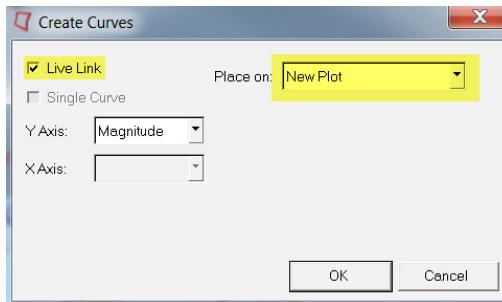


Lets plot the Force vs. Displacement curve (simulation results).

While being inside the Measure Panel  activate the „Create Curves“ button



which opens ...



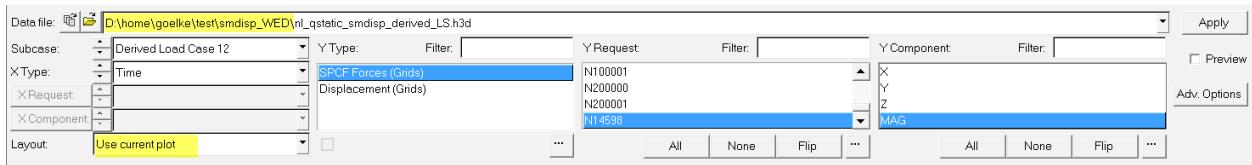
(in here we activate „Live Link“ – quite helpful if we change something in the model later on as it will update the curve automatically)



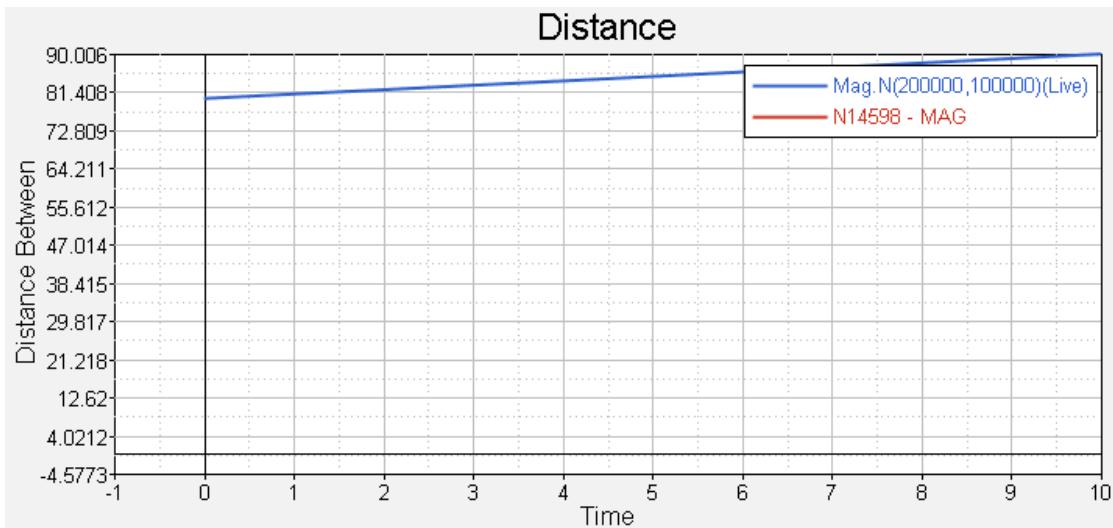
Time is not a physical time. It corresponds to the loadsteps/subcase i.e. we plot elongation per loadstep in one curve.

Click on the graph will open up the HyperGraph dialog window. As data file we select the previously exported *.h3d file which contains the derived load step.

Next we add the Force vs. Time curve to the same plot (in the lower left edge select: Use current plot). Recall: Node 145598 is the Master Node of the RBE2 element.



That's what we get:



Obviously, we need to add another y-axis (Force) and to adjust the scaling of the axes, respectively. Activate



Axis: Y1 Vertical

Primary

Y1

Label: Force

Axis: Y1 Vertical

Primary

Y1

Linear

Logarithmic

dB20

dB10

Min: Auto format

Max: Tics per axis

Auto fit

Fit range

Grids per tic:

Precision:

However, we also need to assign the axes to the corresponding curve



Curves:

Mag.N(200000,100000)(Live)

N14598 - MAG

All
None
Flip
Displayed

Scale and offsets:

X scale:

Y offset:

Y scale:

Y offset:

Move to:

X:

Y:

Axis Assignment:

X: Primary

Y: Primary

Show Manipulator

Curves:

Mag.N(200000,100000)(Live)

N14598 - MAG

All
None
Flip
Displayed

Scale and offsets:

X scale:

X offset:

Y scale:

Y offset:

Move to:

X:

Y:

Axis Assignment:

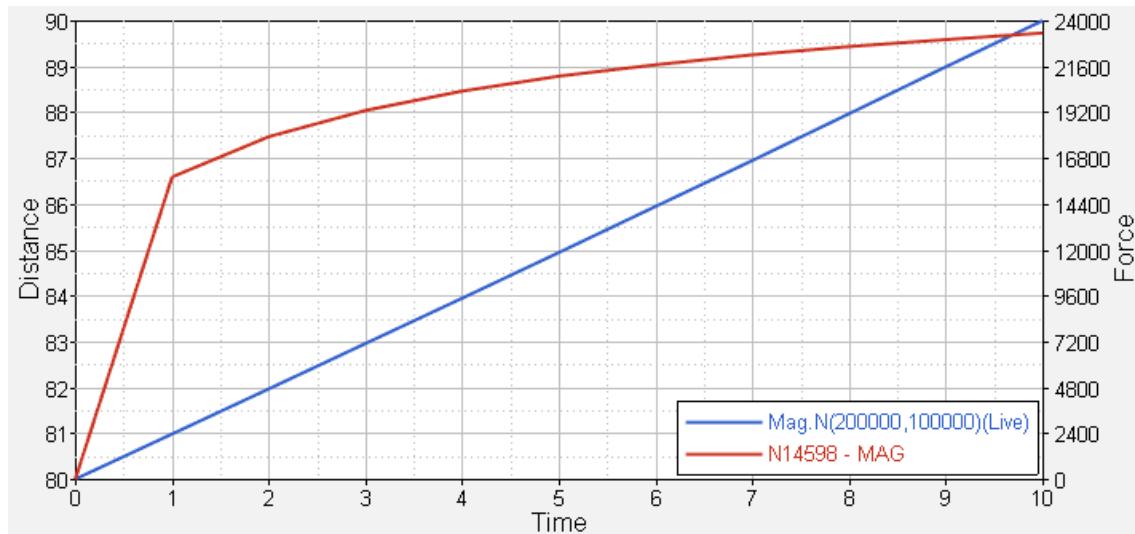
X: Primary

Y: Y1

Show Manipulator

with Y1 = Force axis.

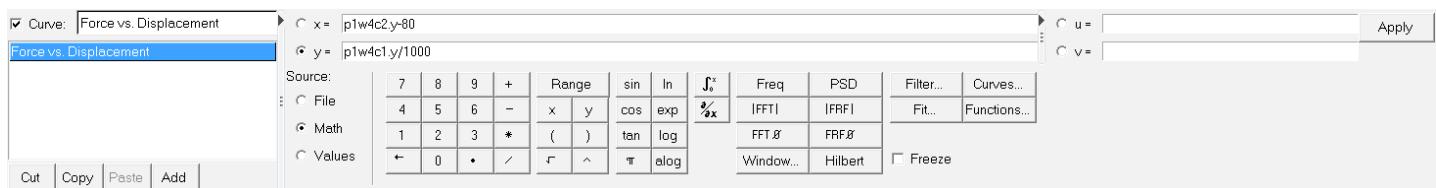
Eventually, the curves (distance/elongation and force) are as:



Mag. N refers to the distance between the nodes with the ID's 200000 and 100000. N14598 is the force at the this particular node

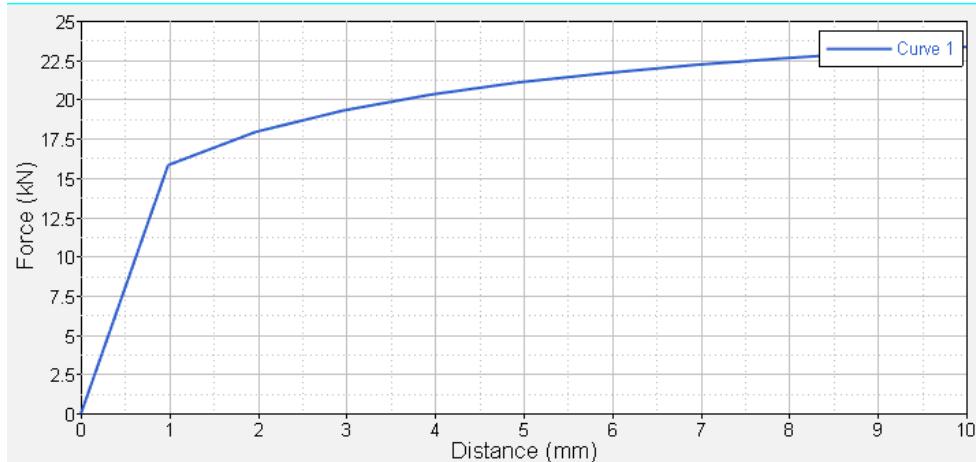
What we need, however, is the graph Force vs. Distance. This graph will now be derived from the plot above:

We add another window to the layout and create the curve of interest

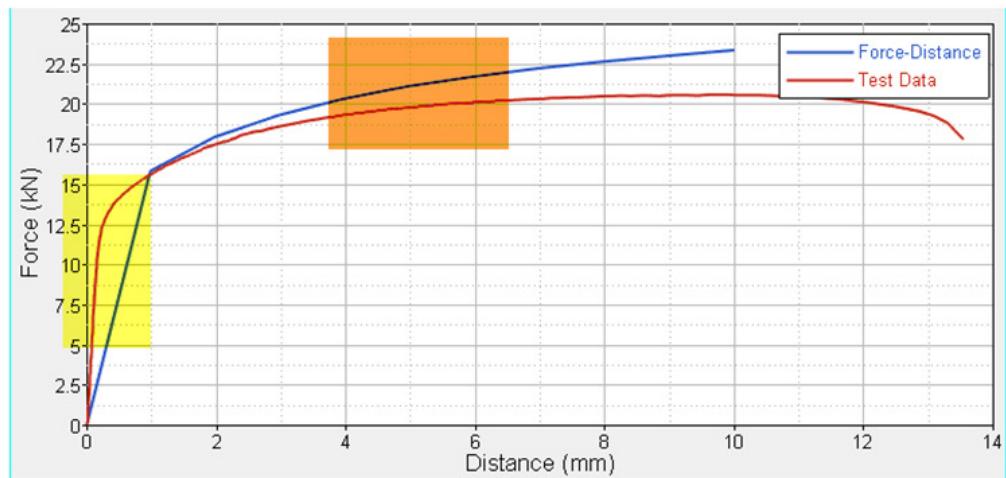


Note the syntax: p = page (should be 1 on your end as well); w = window number; c.y = curve number and its y value (i.e. the y-value of the distance curve in window 4 will be plotted as x-axis in the new graph). Here we subtract 80 mm as the initial distance between node 200000 and 100000 was 80 mm.

The force will be plotted as y-value (derived from page 1, window 4, curve 1 and its respective y value). The force will be displayed as kN, hence we divide the force by factor 1000.



Finally, overlay the test data from the lab (test_results.csv)



Apparently, there is a striking difference between modeling results and test data for elongations of the sample of up to 1 mm. However, note that in the yellow marked area we just have 2 data points: subcase 1 and subcase 2 (linear connected). Hence, in order to fit the red (test) curve better, more subcases are needed. Overall, the model curve consists of 10 data pairs whereas the test data consists of 213 data pairs.

The situation in the orange marked area is different. Here the model data don't follow the test curve.

Why? Any ideas?

Recall the warning message

*** Nonlinear solution has converged ***

*** WARNING # 3200
Maximum plastic strain in this solution: 0.103 exceeds the limit of small deformation theory. The validity of this solution is questionable.

Keep in mind that small displacement theory implies that the cross-sectional properties of the sample remain constant despite the applied loads.

Hence we need to switch from small displacement to large displacements theory.

This will be discussed in the following (based on the OptiStruct Help documentation).

Large displacement nonlinear static analysis is used for the solution of problems wherein the load-response relationship is nonlinear and structural large displacements are involved. The source of this nonlinearity can be attributed to multiple system properties, for example, materials, geometry, nonlinear loading and constraint. Currently, in OptiStruct the following large displacement nonlinear capabilities are available, including large strain elasto-plasticity, hyperelasticity of polynomial form, contact with small tangential motion, and rigid body constraints.

Nonlinear problems are generally history dependent. In order to achieve a certain level of accuracy, the solution must be obtained in a series of small increments. For this purpose we need to solve the equilibrium equation at each increment and a corresponding increment size is selected.

Newton's method is used to solve the nonlinear equilibrium equation in OptiStruct. If the solution is smooth, quadratic of rate of convergence may be achieved when compared with other methods. This method is also very robust in highly nonlinear situations.

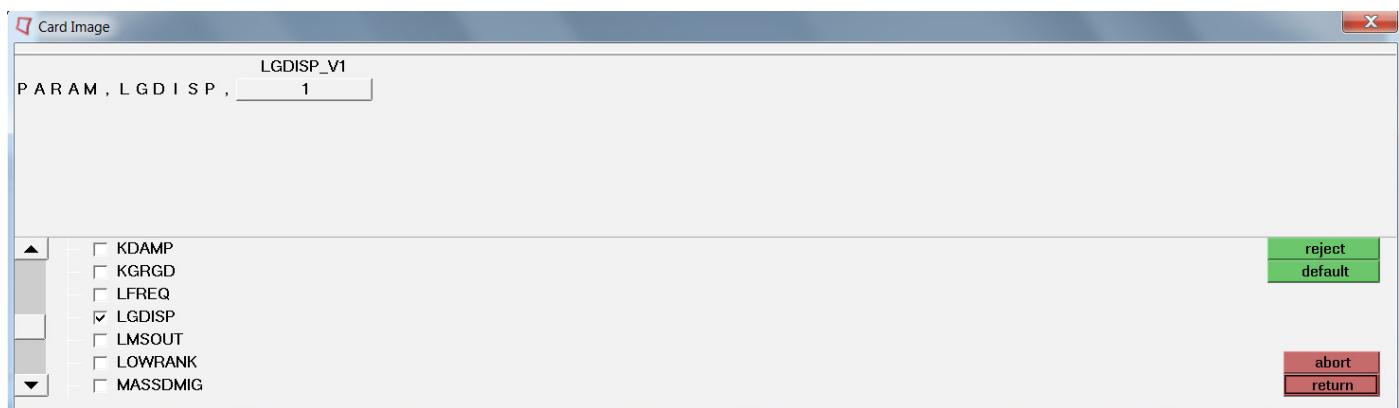
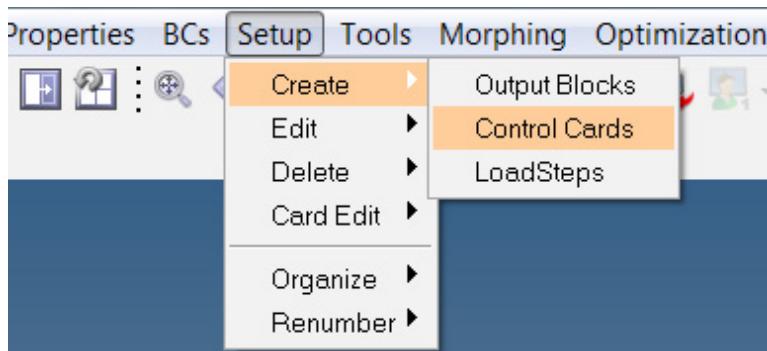
Choosing a suitable time increment is very important. In OptiStruct, an automatic time increment control is available. It should be suitable for a wide range of nonlinear problems and, in general, is a very reliable approach.

The automatic time increment control functionality measures the difficulty of convergence at the current increment. If the calculated number of iterations is equal to the optimal number of iterations for convergence, OptiStruct will proceed with the same increment size. If a lesser number of iterations is required to achieve convergence, the increment size will be increased for the next increment. Similarly, if it is determined that too many iterations are required, the current increment will be attempted again with a smaller increment size.

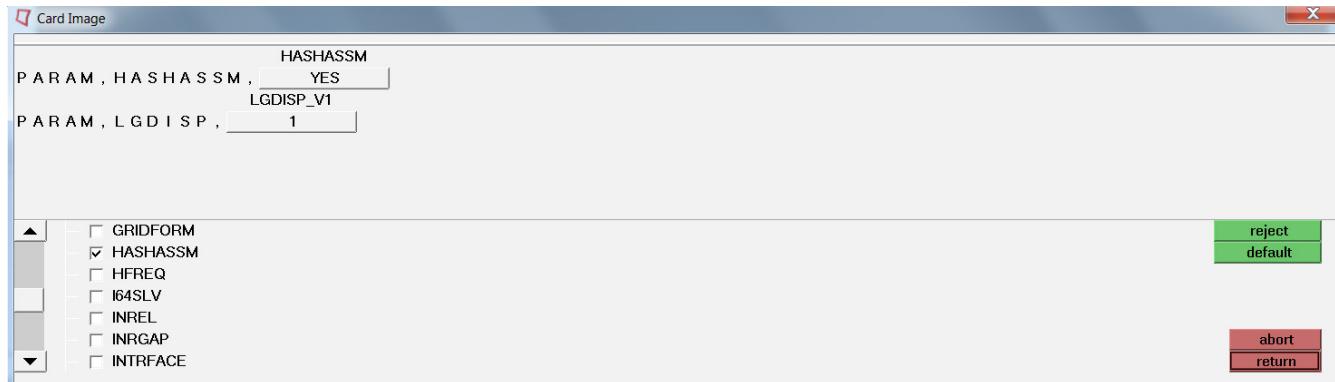
Nonlinear Large Displacement Analysis Problem Setup

The setup for the large displacement nonlinear static solution is straightforward.

- PARAM, LGDISP,1 is used to activate large displacement analysis.



- To indicate that a nonlinear solution is required for any subcase, the NLPARM subcase information entry should be included in the corresponding subcase (as before)
- This subcase entry, in turn, references a NLPARM bulk data entry that contains the convergence tolerances and other nonlinear parameters (as before)
- If constraints or contacts are defined in the model, the matrix profile may be updated over time, so it is recommended that hash assembly is used for nonlinear analysis. This is activated using PARAM, HASHASSM,1.



e. The material MATS1 (TYPE=PLASTIC) is required in conjunction with PARAM, LGDISP,1 to activate large strain elasto-plasticity analysis.

Note: Large displacement nonlinear analysis is supported only for solid elements, RROD, RBAR, RBE2, and RBE3 entries. The expert system (PARAM, EXPERTNL) is currently not supported with large displacement nonlinear analysis.

Large Displacement Nonlinear Analysis is not supported in conjunction with the following elements:

- The following elements can exist in the model, but they will be resolved using small-displacement nonlinear theory (equivalent to ANALYSIS = NLSTAT with plasticity). If large translations or rotations are expected on these elements, then the results may not be accurate:

Shell, Gasket, and Bushing elements

- The following elements are not allowed and OptiStruct will error out if they are present:

CBAR, CBEAM, CGAP, CGAPG, CWELD, CSEAM, CFAST, RBE1, CROD, CELAS, and CONM

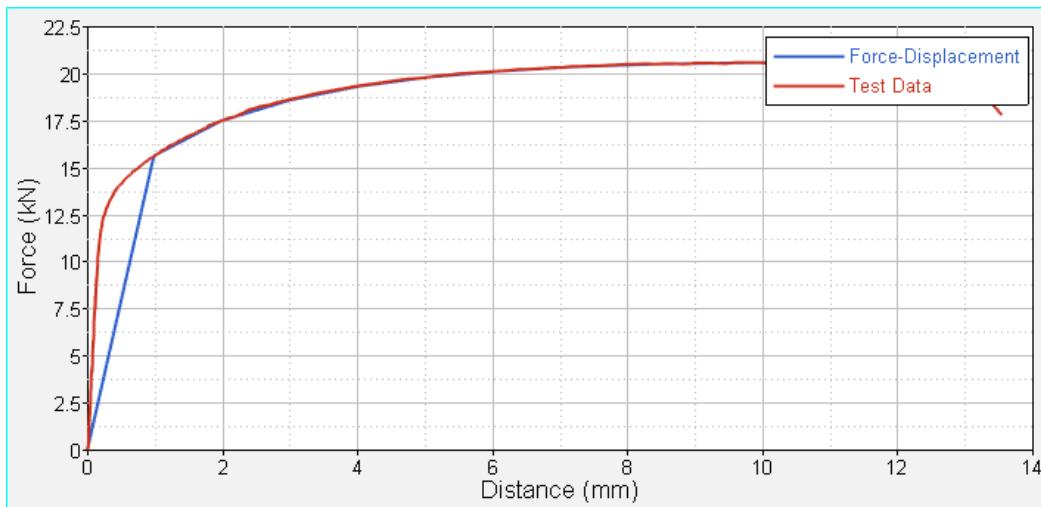
After the above mentioned „additons“ have been made to the model

Name	Value
FFRSLFREQ	
FFRSNCPU	
FLEXH3D	
FLPOK	
FRIC	
FZERO	
G	
GAPOFFST	
GE_MOD	
GENFILE	
GFL	
GMAR	
GMARI	
GRDPNT	
GRIDFORM	
HASHASSM	YES
HREQ	
I64SLV	
INREL	
INRGAP	
INTRFACE	
ITAPE	
K4CUTOFF	
KDAMP	
KGRGD	
LFREQ	
LGDISP	1
LMSOUT	
LOWRANK	
MASSDMIG	
MAXDAMP	

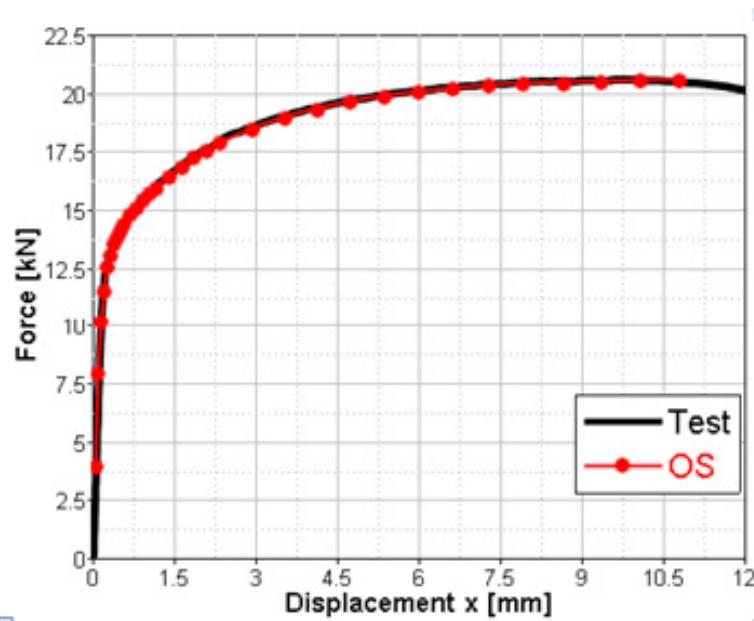
We are ready to launch the NLSTAT analysis.

Postprocessing follows the procedure from before (i.e. create a derived loadcase, save this information as a new h3d file; define a measure between node ID 200000 and 100000 and create curves in HyperGraph)

Eventually, we see that the Force-Displacement curve (simulation) fits nicely the Test Data.



Of course, since we didn't add more loadsteps the differences between both curves for elongations of up to 1 mm remain. The image below is based on a model with 34 loadsteps (versus 10 loadsteps in here).



We recommend - as an additional exercise - to add more loadsteps to your model.

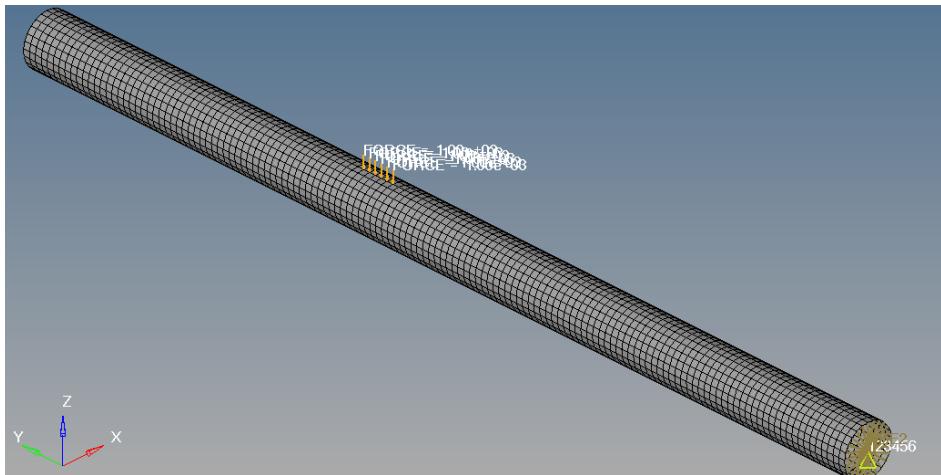
At the end of this chapter about nonlinear material behaviour we would like to challenge you with a question asked in our

“What Do You Think?”

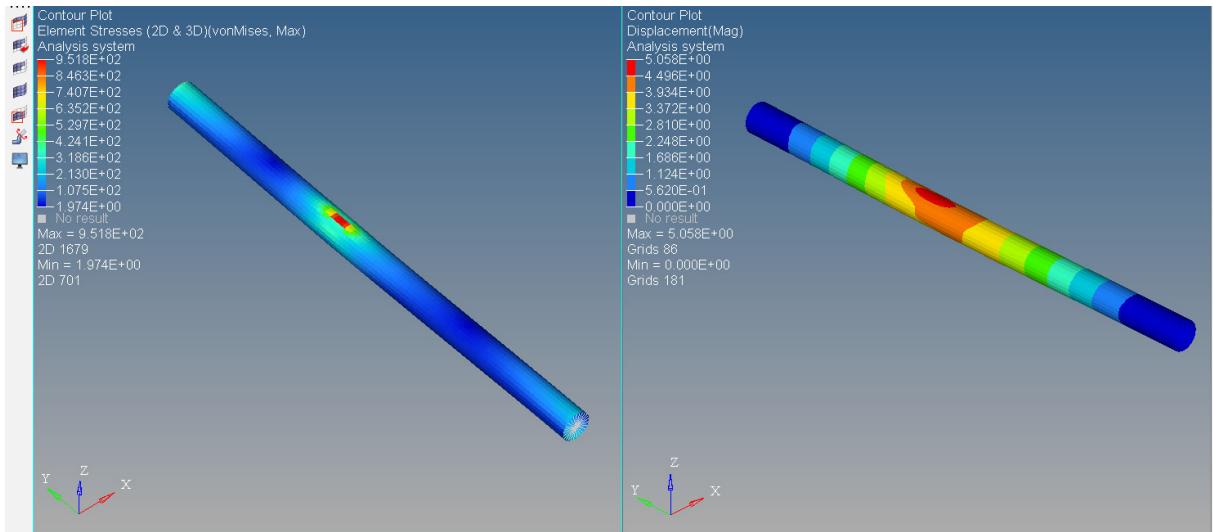
series (in this series we challenge you to find and solve model/simulation errors).

Problem: High stress values

A student was asked to run a finite element analysis on a hollow bar of 1.5mm fixed at both ends and the load applied is 6000 MPa. Aluminium alloy is the material used for the analysis



The student ran a linear static analysis and following are the results: Displacement: 5.058 mm and stress found during post processing is 951.8MPa



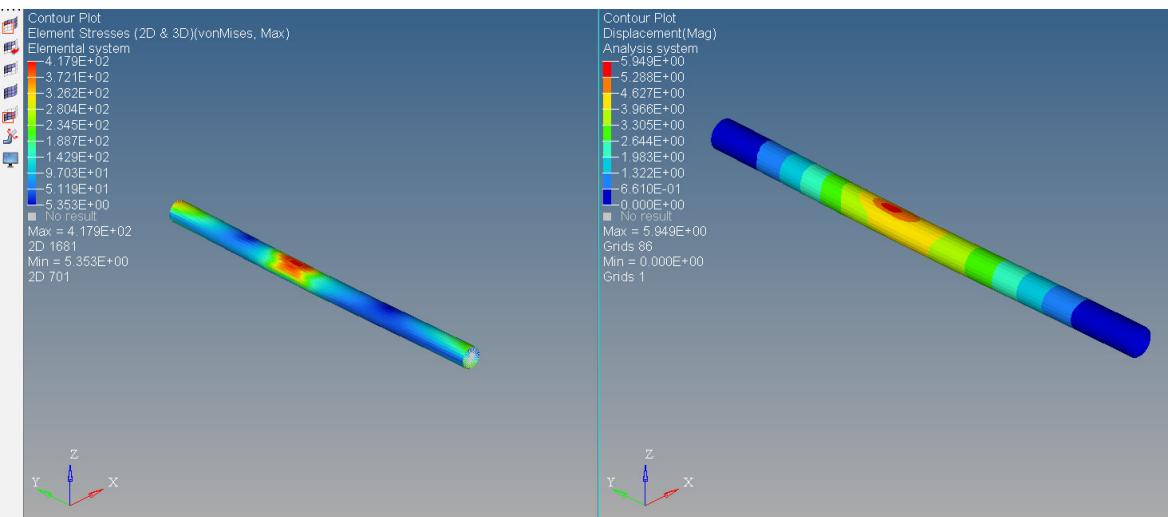
While the yield limit of the material is 350MPa, results show very high stress. Can the user go with these results and accept the design?

What do you Think?

What would have went wrong while modelling?

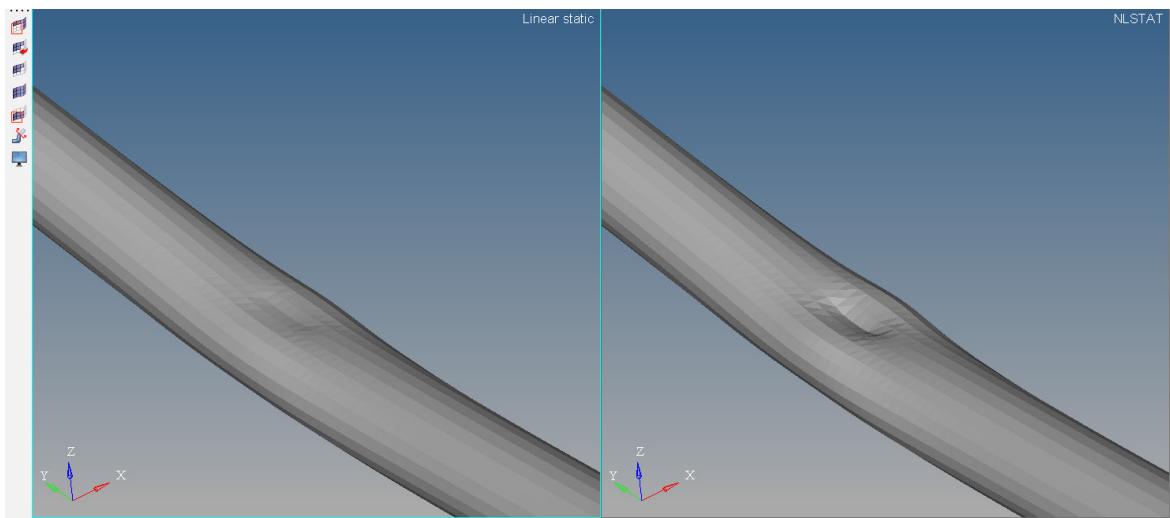
When the stress results are very high and beyond the yield then it is a good idea to consider material nonlinearity. If the loads are high that may cause the structure to deform permanently or if the material is behaving plastically (like rubbers, elastomers, etc...) and if the results required are of good precision then material nonlinearity helps understand the results better.

The below results are from a quasi-static analysis with material curve defined in the properties:



Left: Stress (MPa); right: Displacements (mm)

The deformation is also different from Linear static (Deformation scaled to 5x)



Left: Displacements (mm) linear elastic material; Right: non-linear elastic material (NLSTAT)

More “**What Do You Think?**” challenges are available at:

<http://www.altairuniversity.com/learning-hyperworks/what-do-you-think/>

19.12 Contact Analysis in OptiStruct 13.0

(modified version; original authors are Kristian Holm and Christian Steenbock, Altair Germany)

The modeling of contact between bodies is rather straight forward and includes basically four steps:

Step 1: Create contact elements on the surface/faces of the bodies which will get in contact (i.e. definition of contact surface)

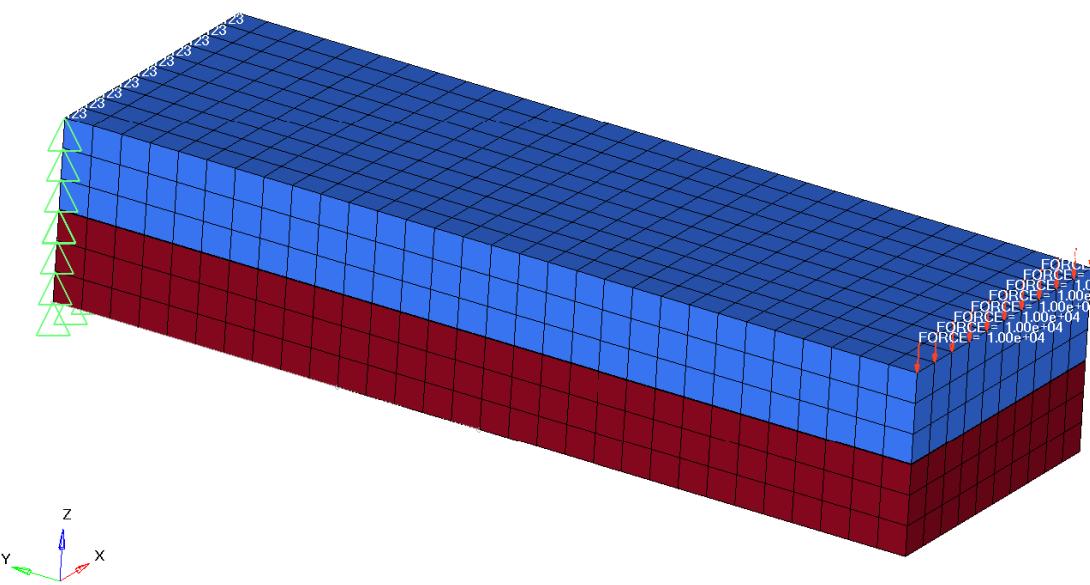
Step 2: Specify which contact surface represents the „Master“ and „Slave“ respectively (=Interface definition). In the same working step (i.e. same panel in HyperMesh) the contact properties such as coefficient of friction are assigned.

Step 3: Create a load collector with Card Image NLPARM to define parameters for the Nonlinear Static Analysis (e.g. number of implicit load sub-increments)

Step 4: Create a load step of type „non-linear quasi-static“

In the following the respective working steps and parameters will be discussed by modeling the contact between two blocks which are separated by a small gap.

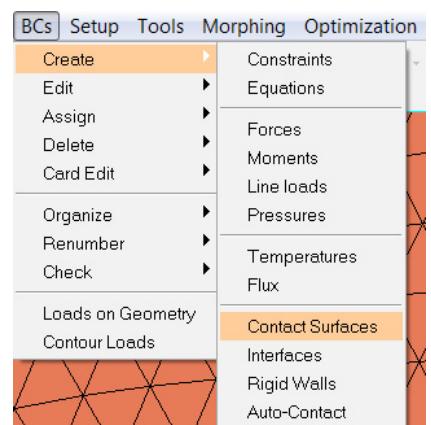
Model length = 300 mm, width = 100 mm, height = 60.25 mm, initial gap between upper and lower block 0.25 mm, element size 10 mm, standard material properties of steel; BC's as shown



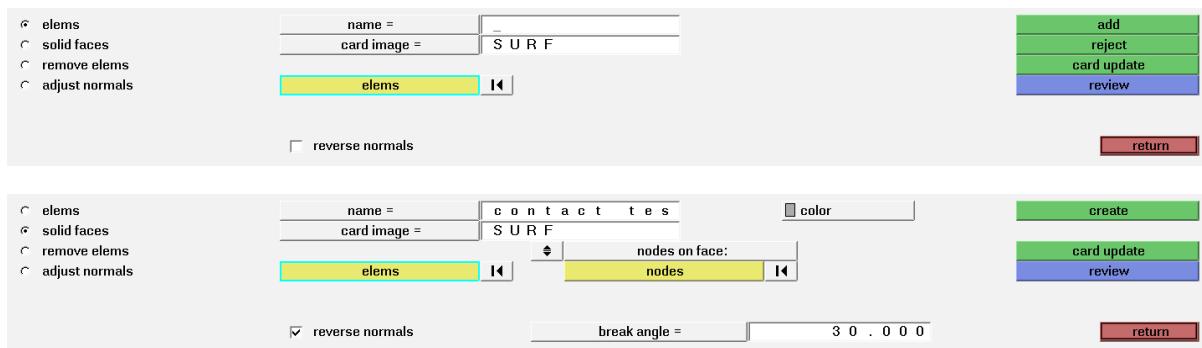
Let's have a look at the individual steps.

Step 1: Contact Surfaces

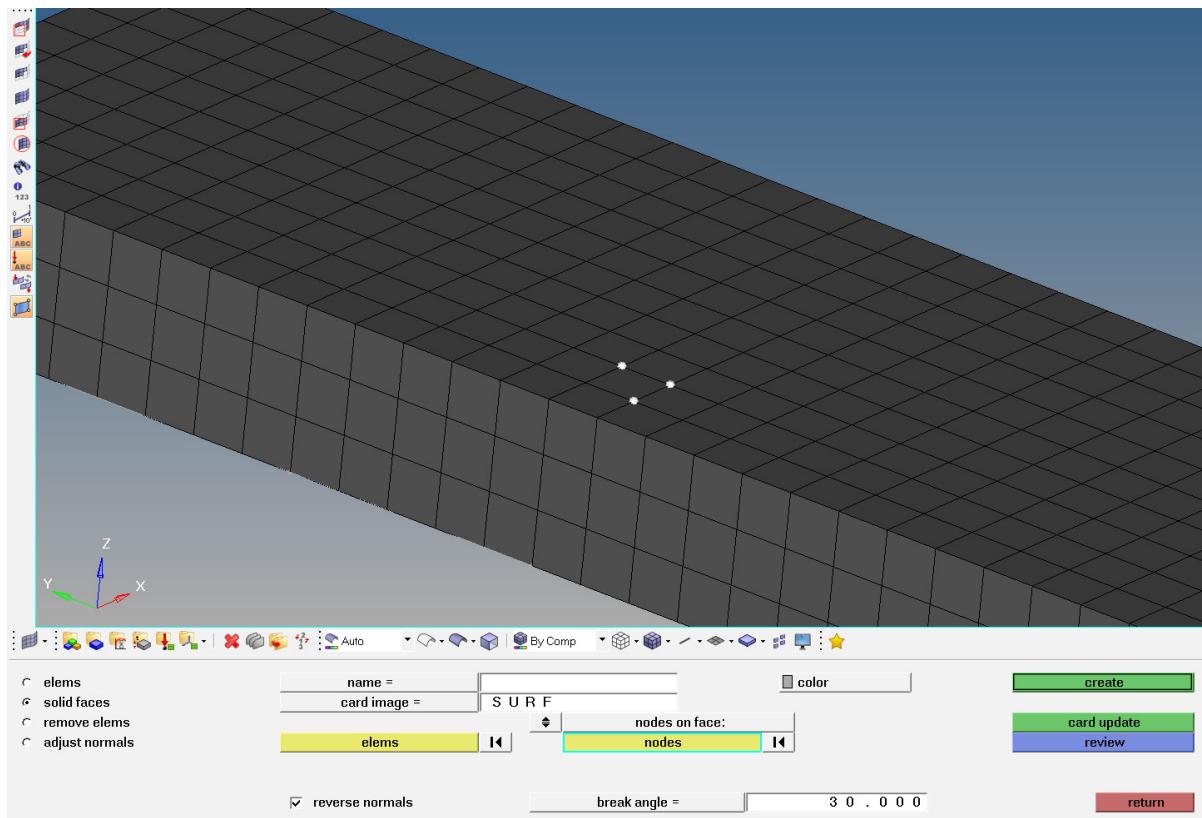
We start by creating Contact Surfaces. Even though it is named “surfaces” it doesn't necessarily mean that we are dealing with a physical surface (CAD).



Depending on whether the contact partners are 2D or 3D elements you may use the „elems“ or the „solid faces“ option. The latter is used for 3D elements. The Card Image is SURF (no other card image listed).

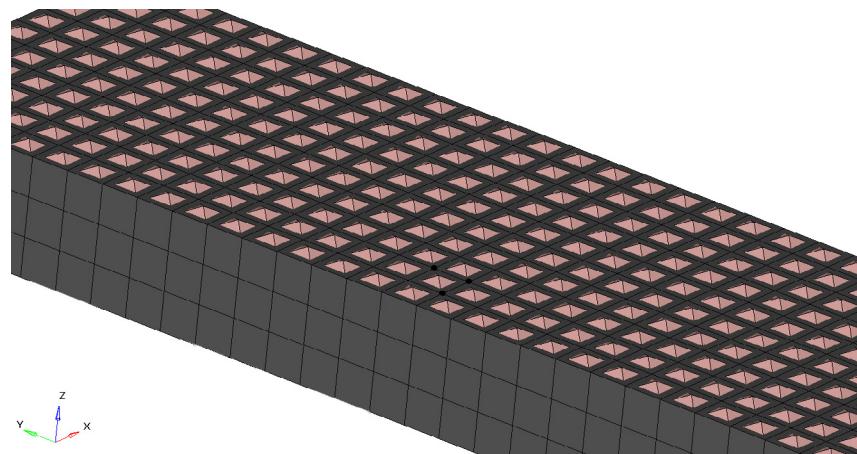


Simply select all 3D elements of for instance the lower block (even the ones which will not be part of the contact pair). Then activate the „nodes on face“ option and select 3 nodes on the contact area (see image below). By adjusting the „break angle“ you can control the sensitivity of the selection algorithm. i.e. the smaller the break angle the more restrictive is the search.

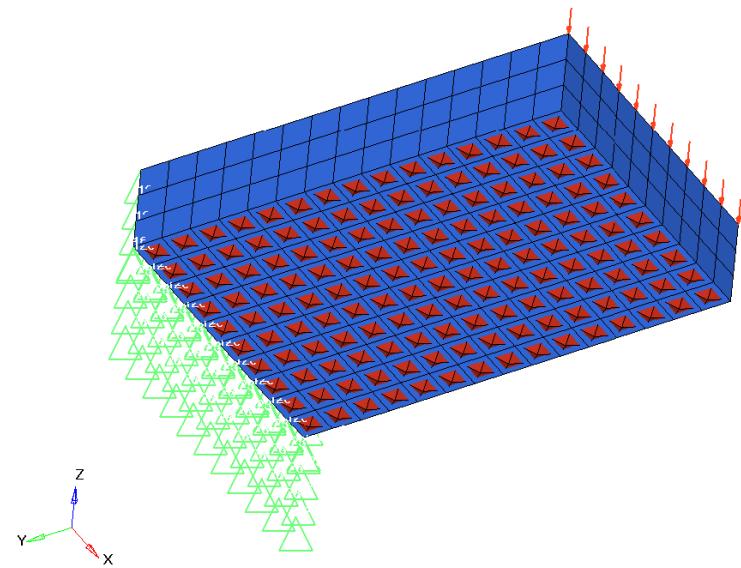


After hitting the create button the contact elements are created.

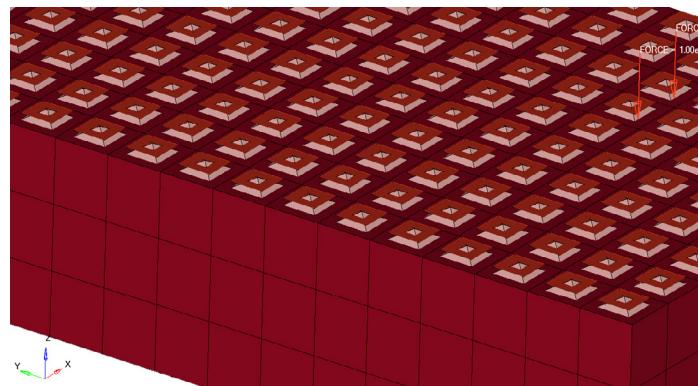
The same process needs to be repeated for the other contact side.



Contact elements of lower block



Contact elements of upper block



Contact elements of upper and lower block

Tip: You may add a _M or _S to the contact surface name to identify Master and Slave contact.

Tip: You may have to make sure that the contact normals are facing each other. Otherwise set the contact normals as if they are facing each other using „adjust normals“ in same panel.

Step 2: Contact Interface Definition

In the next step the contact properties (contact interface) will be defined via the CONTACT Card

The Contact Card defines the contact interface for Nonlinear Quasi-Static Analysis (NLSTAT), Nonlinear Transient Dynamics (NLGEOM), and Contact-based Thermal Analysis (HEAT).

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
CONTACT	CTID	PID/ TYPE/ MU1	SSID	MSID	MORIENT	SRCHDIS	ADJUST	CLEARANCE	
	DISCRET								

The screenshot shows a CAD software interface with a top navigation bar: BCs, Setup, Tools, Morphing, Optimization. A context menu is open over a 3D model of a mechanical part. The 'Create' and 'Interfaces' options are highlighted in orange. Below the interface, a card editor window displays the following card:

```

CONTACT
DISCRET

```

Select CONTACT, provide a name for the contact and hit „create/edit“.

The CONTACT card is then defined as

CONTACT	CTID DISCRET	TYPE SLIDE	SSID ERROR	MSID ERROR	MORIENT	[SRCHDIS]	ADJUST
---------	-----------------	---------------	---------------	---------------	---------	-----------	--------

User Comments
▼ Hide In Menu/Export

Property Option
▼ Property Type

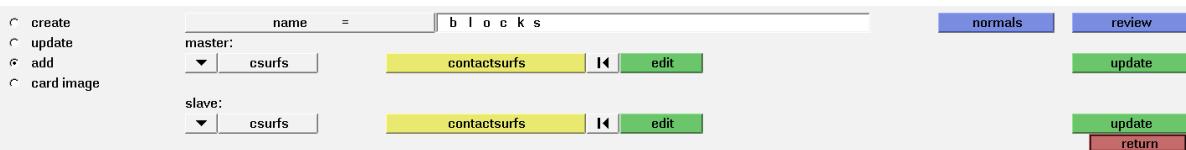
Adjust Option
▼ String Value

reject
default

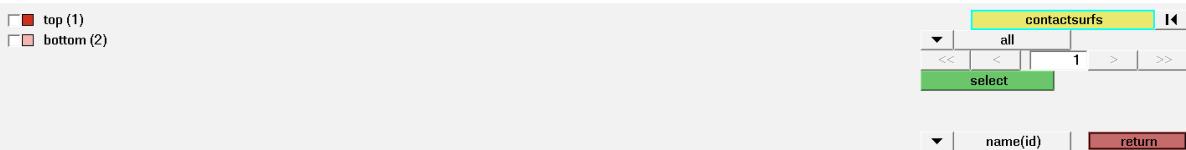
abort
return

The Slave Surface and Master Surface (SSID, MSID) haven't been referenced yet, which is marked by the ERROR message respectively.

Hence, we need to add the missing contact surface information (same panel)



Clicking on „contactsurfs“ next to master opens the selection of available Contact Surfaces (the ones previously created)



Don't forget to hit the update panel **update** – otherwise your work (selection) will not be updated.

Update the slave „contactsurfs“ accordingly.

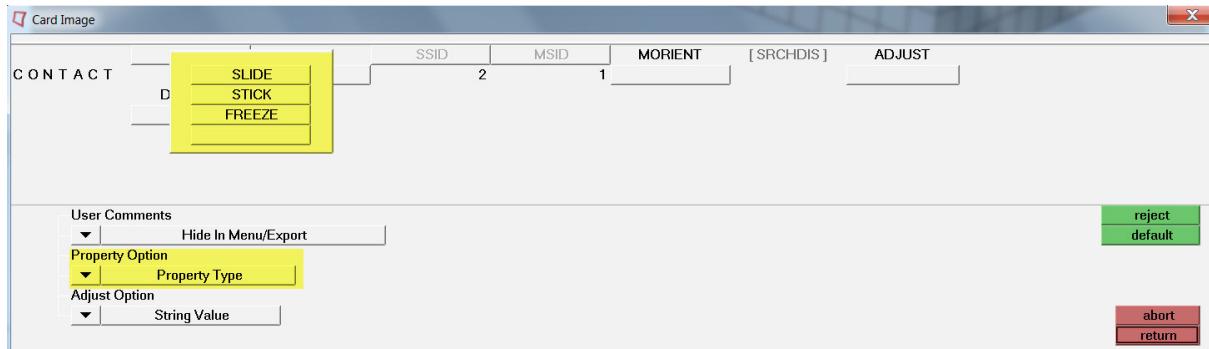
Checking the Card Image of the Interface definition again reveals:

CONTACT	CTID	TYPE	SSID	MSID	MORIENT	[SRCHDIS]	ADJUST
DISCRET	1	SLIDE	2	1			

User Comments	Hide In Menu/Export	reject default
Property Option	Property Type	abort return
Adjust Option	String Value	

Slave refers to the contact surface named „bottom“ with the ID 2, Master is named „top“ with ID 1.

Note that by default, TYPE offers you to either select SLIDE, STICK or FREEZE.



TYPE:

Choose type of contact without pointing to contact property – respective default property settings will be used. Default settings can be changed using CONTPRM.

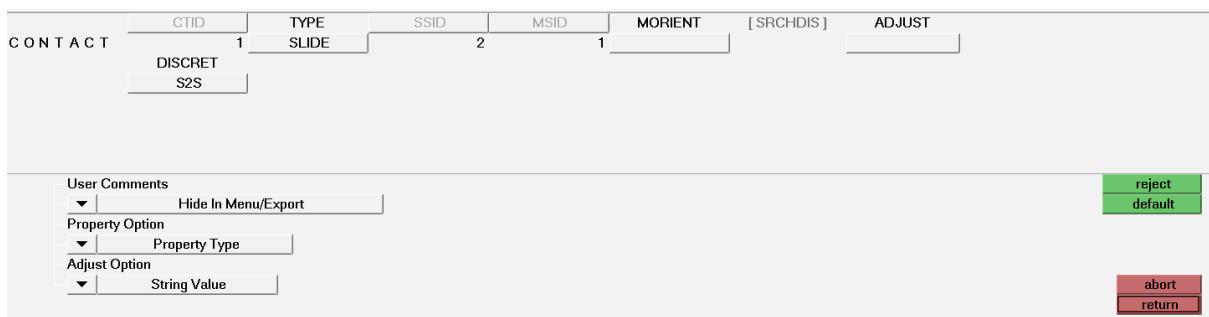
SLIDE – Sliding contact. If nothing else is specified, it is assumed that coefficient of friction is „0“.

STICK – Contact with stick condition (stick applies to closed contacts only). TYPE=STICK is interpreted in OptiStruct as an enforced stick condition - such contact interfaces will not enter the sliding phase. Of course, the enforced stick only applies to contacts that are closed.

FREEZE – Enforced zero relative displacements on the contact interface (applies to both closed and open contacts). TYPE=FREEZE

enforces zero relative motion on the contact surface – the contact gap opening remains fixed at the original value and the sliding distance is forced to be zero. Also, rotations at the slave node are matched to the rotations of the master patch. The FREEZE condition applies to all respective contact elements, no matter whether open or closed.

Here the CONTACT definition is using TYPE = SLIDE. Note, as no friction is defined, the interface is frictionless.



The DISCRET parameter refers to the Discrete Discretization approach type for the construction of contact elements.

Two options/technologies are available

N2S – node-to-surface discretization (default)

S2S – surface-to-surface discretization

DISCRET = N2S is recommended if the slave entity is a set of grids (nodes) or a set of solid elements.

MORIENT

Orientation of contact “pushout” force from master surface. Applies only to masters that consist of shell elements or patches of grids. Masters defined on solid elements always push outwards irrespective of this flag.

OPENGAP – The contact interface is assumed open.

OVERLAP – Slave and master bodies overlap.

NORM – Contact force is oriented along the vector normal to the master surface.

REVNORM – Contact force is oriented opposite to the default vector normal to the master surface.

Default = OPENGAP (i.e. leave it blank)

ADJUST

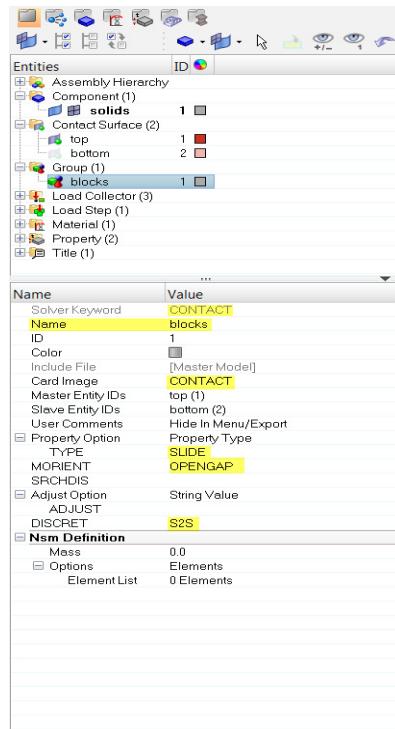
Adjustment of slave nodes onto the master surface at the start of a simulation.

Default setting = NO - no adjustment (leave it blank).

AUTO – A real value equal to 5% of the average edge length on the master surface is internally assigned as the depth criterion

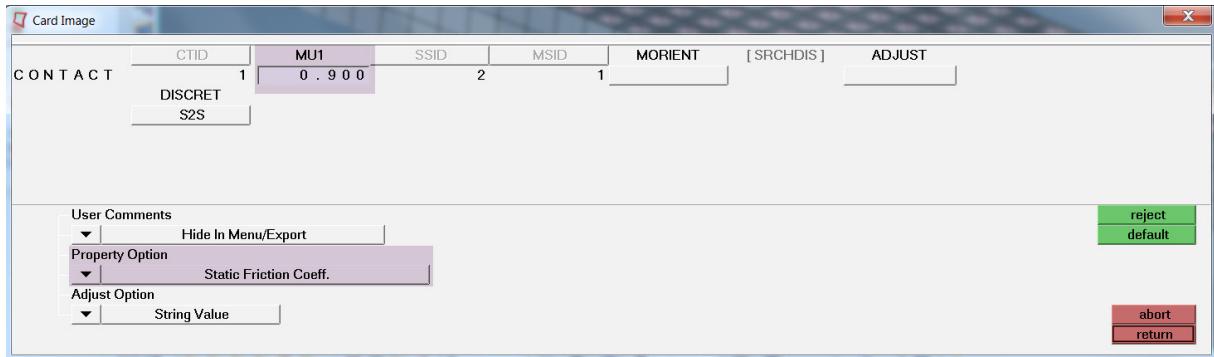
Other options are available too. However, just get started with the default setting (more information about this parameter is included in the Help Documentation → Contact)

The CONTACT card is also displayed in the browser:



Coefficient of Friction

You may specify the Coefficient of Static Friction directly in the CONTACT Card. Just change Property Option to Static Friction Coefficient (μ_s) as shown below.



The Static Friction Coefficient (μ_s ; MU1) directly defined on the CONTACT card allows for simplified specification of frictional contacts. Note that this implies Kinetic Coefficient of Friction (MU2) equals the Static Friction Coefficient MU1=MU2, unless MU2 is specified explicitly on the CONTPRM card (explained further below). Also note that the value of MU1 assigned on the CONTACT card must be less than 1.0 – to specify higher values of static coefficient of friction, PCONT card must be used.

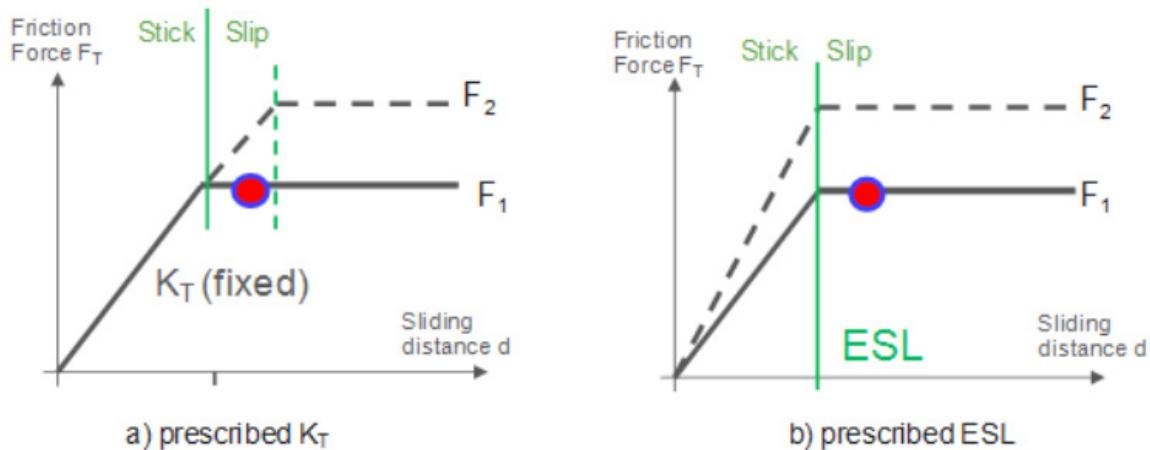
MU1=STICK is interpreted in OptiStruct as an enforced stick condition - such contact interfaces will not enter the sliding phase. Of course, the enforced stick only applies to contacts that are closed

MU1=FREEZE enforces zero relative displacements on the contact surface – the contact gap opening remains fixed at the original value and the sliding distance is zero. The FREEZE condition applies to all slave nodes, no matter whether their initial gap is open or closed.

Effective in Release 12.0, two models of friction are available in nonlinear analysis:

- (a) Model based on fixed slope K_T (previously existing)
- (b) Model based on Elastic Slip Distance FRICESL (introduced in v12.0 and current default).

This latter model typically shows better performance in solution of frictional problems thanks to more stable handling of transitions from stick to slip. Key differences between the two available models are illustrated in the figure below (F_1 and F_2 represent two different values of normal force F_N):



Comparison of the two friction models for contact elements.

Model (a), based on fixed stiffness K_T , is relatively simple, yet has certain drawback in modeling nonlinear friction. Namely, in Coulomb friction the frictional resistance depends upon normal force. Using fixed K_T will predict different range of stick/slip boundary for different normal forces, and thus may qualify the same configuration as stick or slip, depending on normal force.

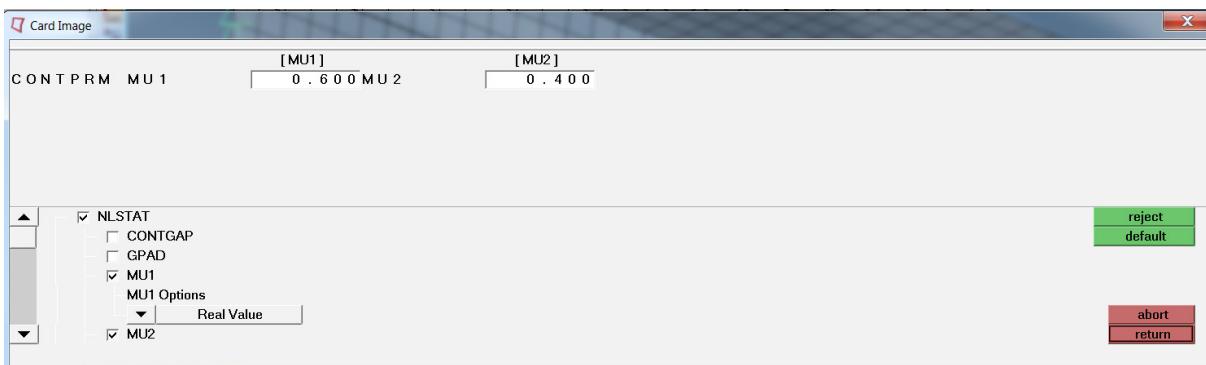
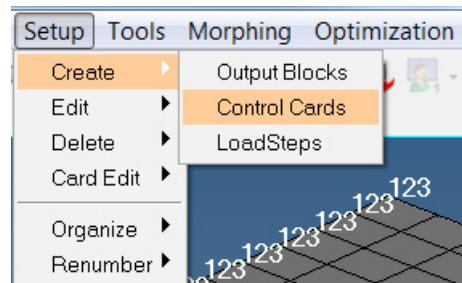
Model (b), based on Elastic Slip Distance, provides unique identification of stick or slip and generally performs better in problem solving with friction. This model does require setting elastic slip distance FRICESL – for contact interfaces this value is determined automatically as 0.5% of typical element size on all Master contact surfaces.

The model (b), which is currently the default, is recommended for solution of nonlinear problems with friction. For backwards compatibility, the model based on fixed K_T can be activated by setting FRICESL=0 on the PCONT or CONTPRM card.

Contact Properties defined on the CONTPRM Card

The CONTPRM Card defines the default properties of all contacts and sets parameters that affect all contacts. The default values set here can be overridden by values explicitly specified on PCONT (we will have a look at PCONT further below) and CONTACT cards.

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
CONTPRM	PARAM1	VALUE1	PARAM2	VALUE2	PARAM3	VALUE3	PARAM4	VALUE4	
	PARAM5	VALUE5							



The coefficient of static friction is arbitrarily set to 0.6, the coefficient of kinetic friction to 0.4. The same information can also be reviewed in the browser:

Name	Value
Solver Keyword	CONTPRM MU1
Include File	[Master Model]
Status	<input checked="" type="checkbox"/>
NLSTAT	<input checked="" type="checkbox"/>
CONTGAP	<input type="checkbox"/>
GPAD	<input type="checkbox"/>
MU1	<input checked="" type="checkbox"/>
MU1 Options	Real Value
MU1	0.6
MU2	<input checked="" type="checkbox"/>
MU2	0.4
STIFF	<input type="checkbox"/>
FRICESL_opts	<input type="checkbox"/>
NLGEOM	<input type="checkbox"/>
CORIENT	<input type="checkbox"/>
UNSUPPORTED_CONTPRMS =	0

There is one other parameter we want to mention here (more parameters are documented in the HyperWorks Help):

Option STIFF=AUTO determines the value of normal stiffness for each contact element using the stiffness of surrounding elements. Additional options SOFT and HARD create respectively softer or harder penalties. SOFT can be used in cases of convergence difficulties and HARD can be used if undesirable penetration is detected in the solution. A negative value for STIFF indicates that a stiffness scaling factor equal to $|Real < 0.0|$ is defined. This scaling is applied on the stiffness value via STIFF = AUTO.

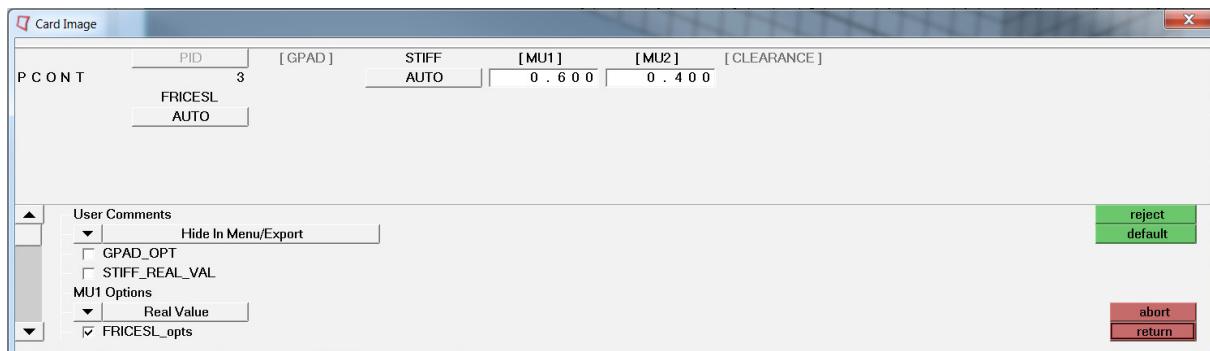
Contact Properties defined on the PCONT Card

Whereas the CONTPRM Card defines the default properties for ALL contacts, the properties of individual contact pairs can be „controlled“ by the PCONT Card (i.e. it overrides CONTPRM)

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
PCONT	PID	GPAD	STIFF	MU1	MU2	CLEARANCE			
	FRICESL								

The PCONT is a property collector. In the example below, the PCONT Card is used to define MU1 (coefficient of static friction), and MU2 (coefficient of kinetic friction).

The PCONT Card Image



Property collector with Card image PCONT

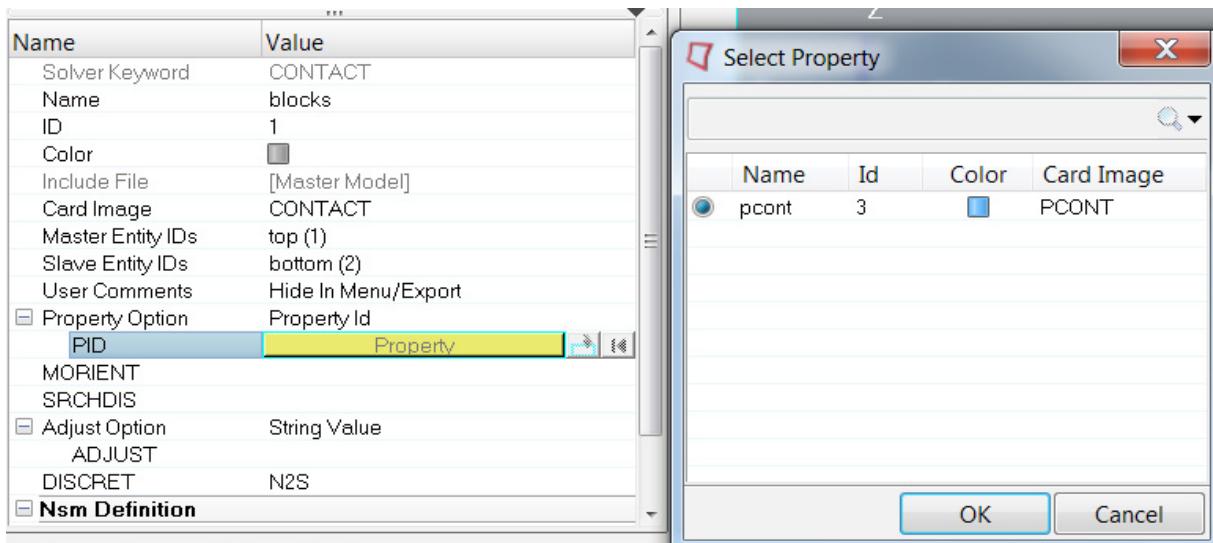
Alternatively, the PCONT property collector may be viewed and referenced within the browser

Name	Value
Solver Keyword	PCONT
Name	pcont
ID	3
Color	■
Include File	[Master Model]
Card Image	PCONT
User Comments	Hide In Menu/Export
GPAD_OPT	GPAD
STIFF_REAL_VAL	AUTO
STIFF	AUTO
MU1 Options	Real Value
MU1	0.6
MU2	0.4
CLEARANCE	
FRICESL_opts	AUTO
FRICESL	AUTO
PCONTX...	
PCONTHT	

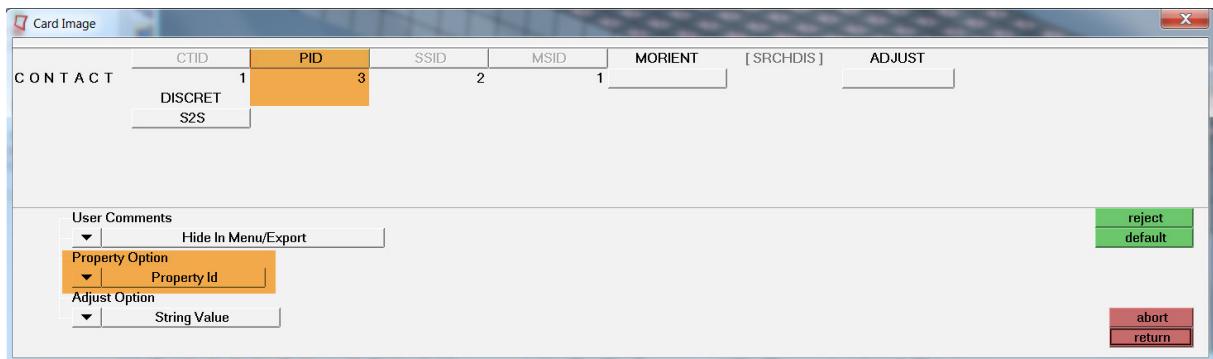
The property collector PCONT may be referenced inside the browser

Name	Value
Color	■
Include File	[Master Model]
Card Image	CONTACT
Master Entity IDs	top (1)
Slave Entity IDs	bottom (2)
User Comments	Hide In Menu/Export
Property Option	Static Friction Coeff.
MU1	Property Type
MORIENT	Property Id
SRCHDIS	Static Friction Coeff.
Adjust Option	String Value
ADJUST	
DISCRET	N2S

Property Option is switched to Property Id, which then allows us to select the property collector.



or (more traditionally) within the Card Editor



Here you need to change the setting under Property Option.

Note: the PCONT Card overwrites the corresponding definitions (if made at all) in the CONTPRM card.

Step 3: Loadcollector of Type NLPARM

Aside the contact definition we need to create an additional load collector with Card Image NLPARM which is used to define parameters for nonlinear static analysis or heat transfer analysis solution control.

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
NLPARM	ID	NINC			KSTEP	MAXITER	CONV		
	EPSU	EPSP	EPSW			MAXLS		LSTOL	

NINC = Number of implicit load sub-increments. Default = 1 (no increments) for ANALYSIS = NLSTAT and ANALYSIS=NLHEAT

Note: The solution method for quasi-static nonlinear analysis (ANALYSIS = NLSTAT) is full Newton. The stiffness matrix is updated at each iteration. NINC > 0 represents the number of equal subdivisions that the total load in a given subcase will be divided into. If NINC is blank, the entire load for a given subcase is applied at once. The Newton method will be applied to consecutive load levels until the final load is reached.

MAXITER: Limit on number of implicit iterations for each load increment. If reached, the solution is terminated (ANALYSIS = NLSTAT and ANALYSIS=NLHEAT). Default = 25

CONV = Flags to select implicit convergence criteria. Default = UPW for ANALYSIS = NLSTAT

EPSU = Error tolerance for displacement (U) criterion. Default = 1.0E-3 for ANALYSIS = NLSTAT

EPSP = Error tolerance for load (P) criterion. Default = 1.0E-3 for ANALYSIS = NLSTAT and ANALYSIS=NLHEAT

EPSW = Error tolerance for work (W) criterion. Default = 1.0E-7 for ANALYSIS = NLSTAT and ANALYSIS=NLHEAT

MAXLS = Maximum number of line searches allowed for each iteration. Default = 0 for ANALYSIS = NLSTAT

The screenshot shows a CAD software interface with two main windows. The top window is the Entity Browser, displaying a tree structure of model entities like Assembly Hierarchy, Components, Contact Surfaces, Groups, Load Collectors, and various steps and properties. The bottom window is a card editor for an NLPARM card, showing a table of parameters and their values.

Name	Value
Solver Keyword	NLParm
Name	nlparm
ID	3
Color	█
Include File	[Master Model]
Card Image	NLParm
User Comments	NLParm
NINC	NSMADD
KSTEP	PEAKOUT
MAXITER	PFAT
CONV	PFPATH
EPSU	PTADD
EPSP	RANDPS
EPSW	RFORCE
MAXLS	RLOAD1
LSTOL	RLOAD2
NLPARMX	█

Name	Value
Solver Keyword	NLParm
Name	nlparm
ID	3
Color	█
Include File	[Master Model]
Card Image	NLParm
User Comments	Hide In Menu/Export
NINC	10
KSTEP	
MAXITER	25
CONV	
EPSU	
EPSP	
EPSW	
MAXLS	
LSTOL	
NLPARMX	█

Step 4: Loadstep Definition

Eventually, we need to create a loadstep/subcase of analysis type „non-linear quasi-static“

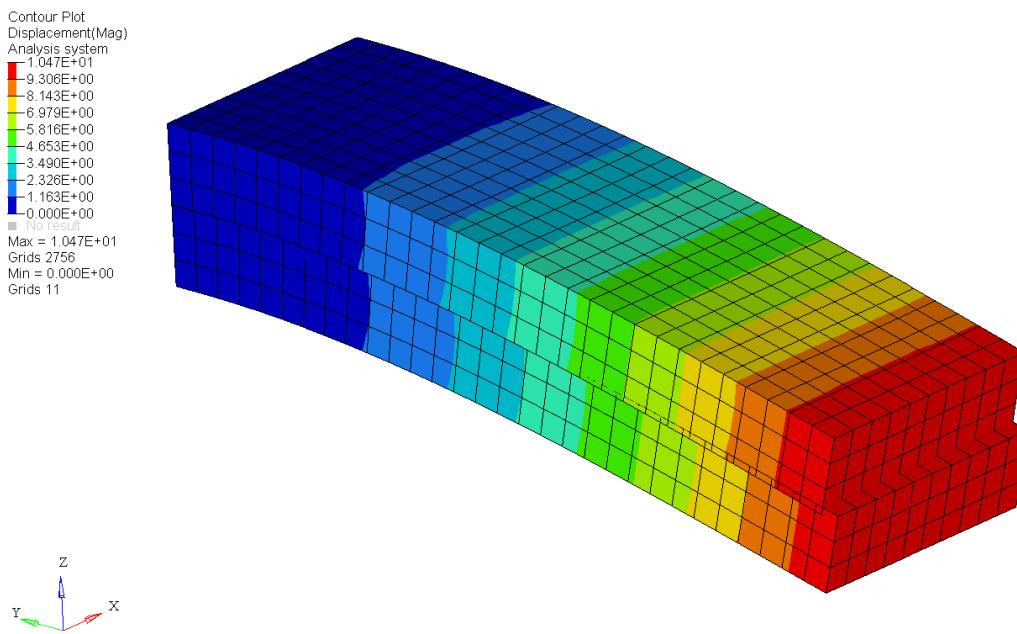
The screenshot shows the Ansys interface with the Entity browser open at the top. Under the 'Load Step (1)' category, 'loadstep' is selected with an ID of 1. Below this, the 'Subcase Definition' dialog box is displayed.

Name	Value
Solver Keyword	SUBCASE
Name	loadstep
ID	1
Include File	[Master Model]
User Comments	Hide In Menu/Export
Subcase Definition	
Analysis type	Non-linear quasi-static
SPC	spc (1)
LOAD	load (2)
SUPPORT	<Unspecified>
DEFORM	<Unspecified>
PRETENSION	<Unspecified>
MPC	<Unspecified>
TEMP	<Unspecified>
NLTERM	nlterm (3)
STATSUB (PRETENS)	<Unspecified>
SUBCASE OPTIONS	
LABEL	<input type="checkbox"/>
SUBTITLE	<input type="checkbox"/>
ANALYSIS	
TYPE	<input checked="" type="checkbox"/> NLSTAT
CNTNLSUB	<input type="checkbox"/>
EIGVRETRIEVE	<input type="checkbox"/>
EIGVSAVE	<input type="checkbox"/>
EXCLUDE	<input type="checkbox"/>
RADSND	<input type="checkbox"/>
RESVEC	<input type="checkbox"/>
SOLVTYP	<input type="checkbox"/>
OUTPUT	<input type="checkbox"/>
SUBCASE_UNSUPPORTED	<input type="checkbox"/>

Save the model as *.hm file and start the analysis.

Postprocessing

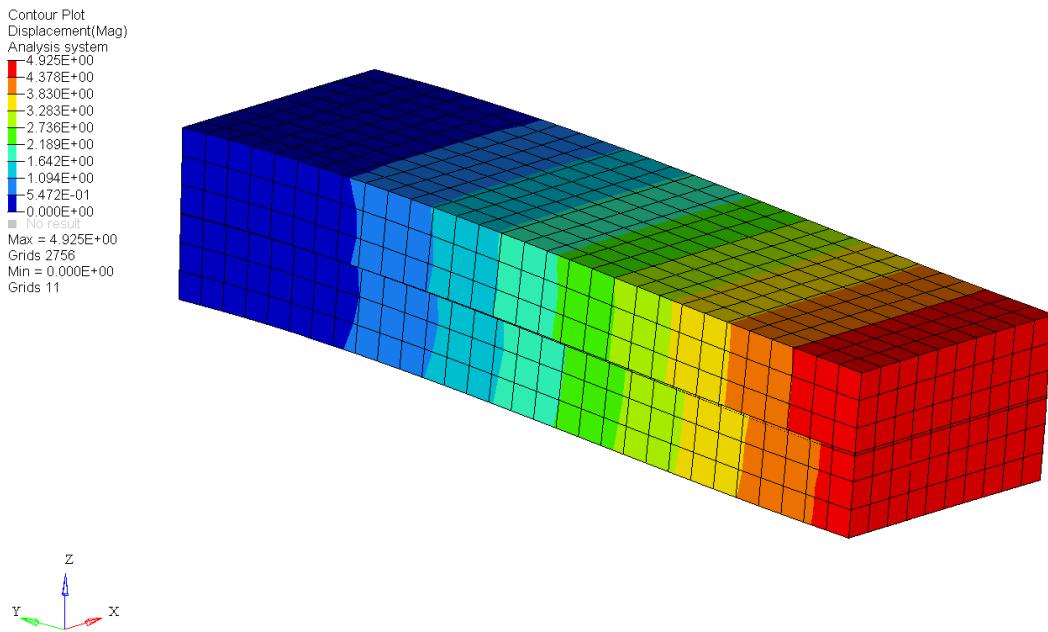
Below we look at a series of simulation results taking into account frictionless contact (SLIDE), contact with friction (SLIDE), STICK, FREEZE and with reversed loads. We start with frictionless contact (SLIDE).



Interface of Type SLIDE (S2S); displacements are scaled by factor 5

In the model below we assume an interface of type STICK (with N2S). The displacements are scaled by factor 5.

Reminder: TYPE=STICK is interpreted in OptiStruct as an enforced stick condition - such contact interfaces will not enter the sliding phase. Of course, the enforced stick only applies to contacts that are closed.

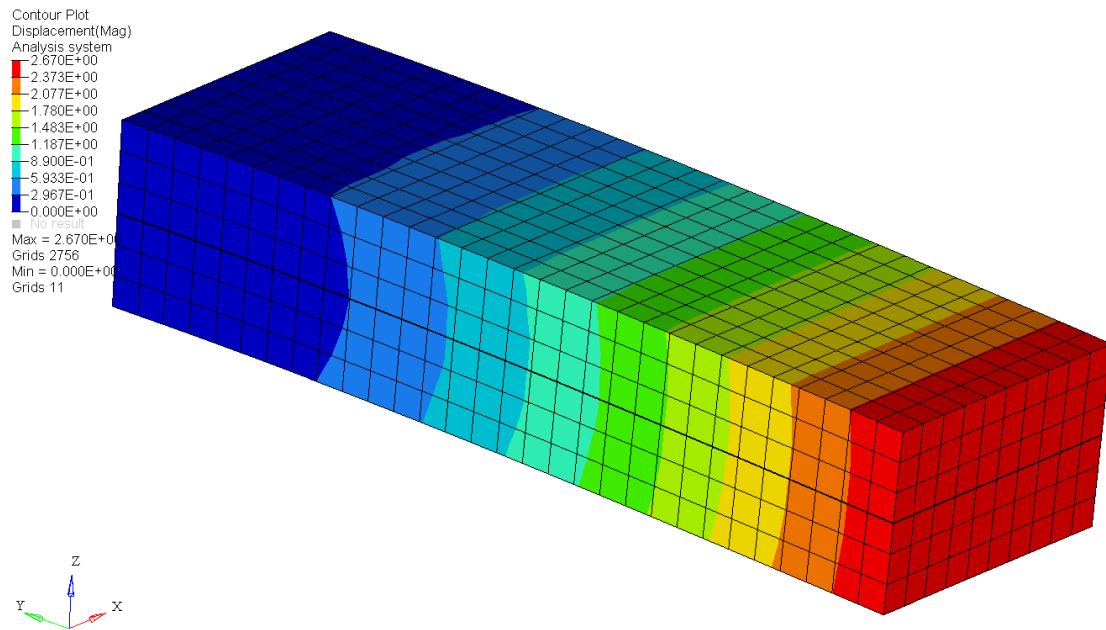


Interface of Type STICK (S2S); displacements are scaled by factor 5

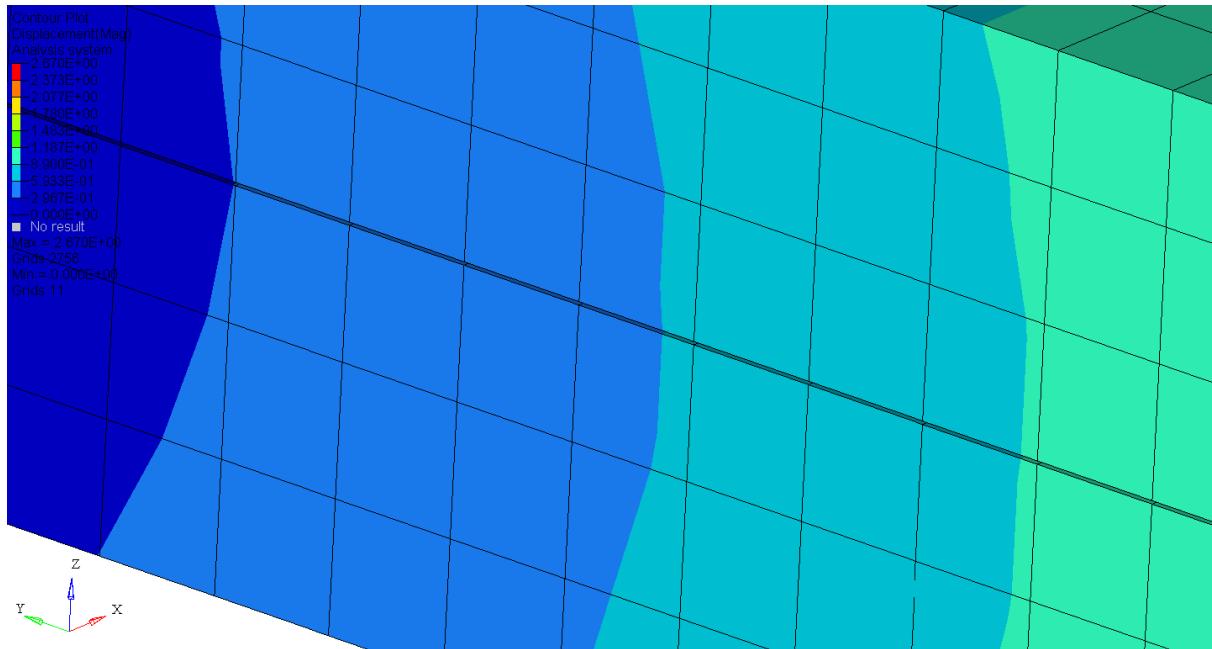
Now we look at an interface of Type FREEZE.

Reminder: TYPE=FREEZE enforces zero relative motion on the contact surface – the contact gap opening remains fixed at the original value and the sliding distance is forced to be zero. Also, rotations at the slave node are matched to the rotations of the master patch. The FREEZE condition applies to all respective contact elements, no matter whether open or closed. FREEZE – Enforced zero relative displacements on the contact interface (applies to both closed and open contacts).

- The contact opening remains fixed at the initial value
- Linear static analysis, no NLPARM or NLSTAT needed.

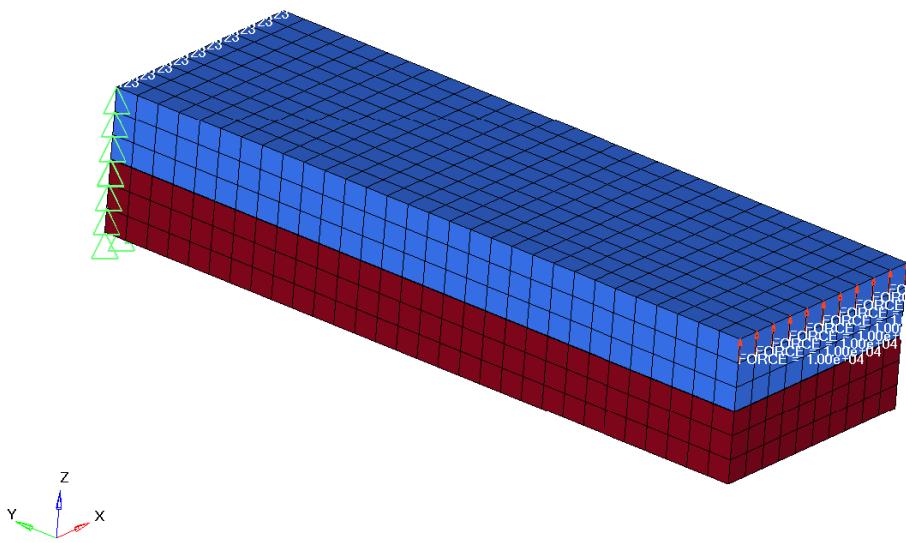


Interface of Type FREEZE (S2S); displacements are scaled by factor 5



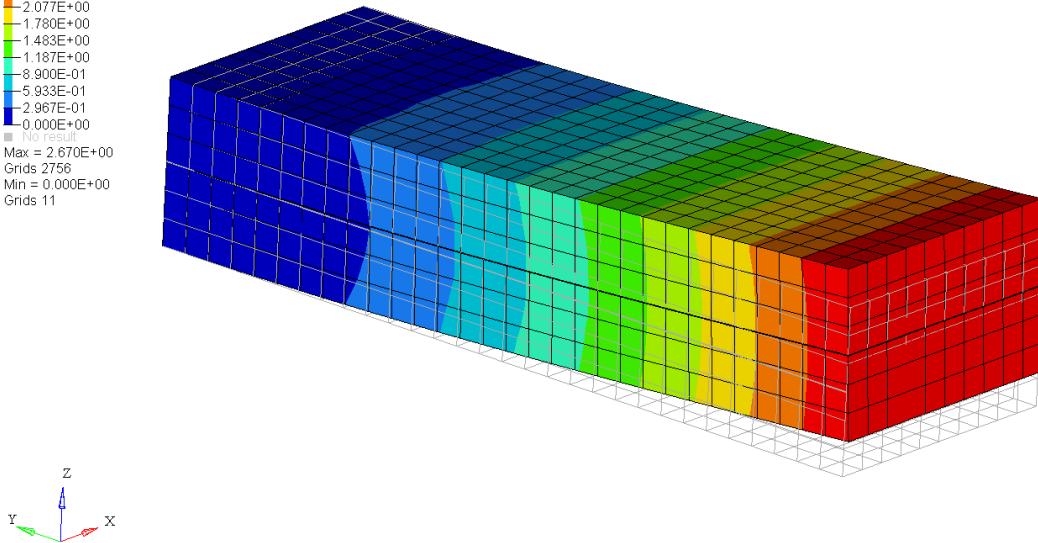
Zoomed in area: the „bold“ line is the interface between the two blocks (gap 0.25 mm). Interface type FREEZE (with N2S); displacements are scaled by factor 5

Opening Blocks – Loads Reversed



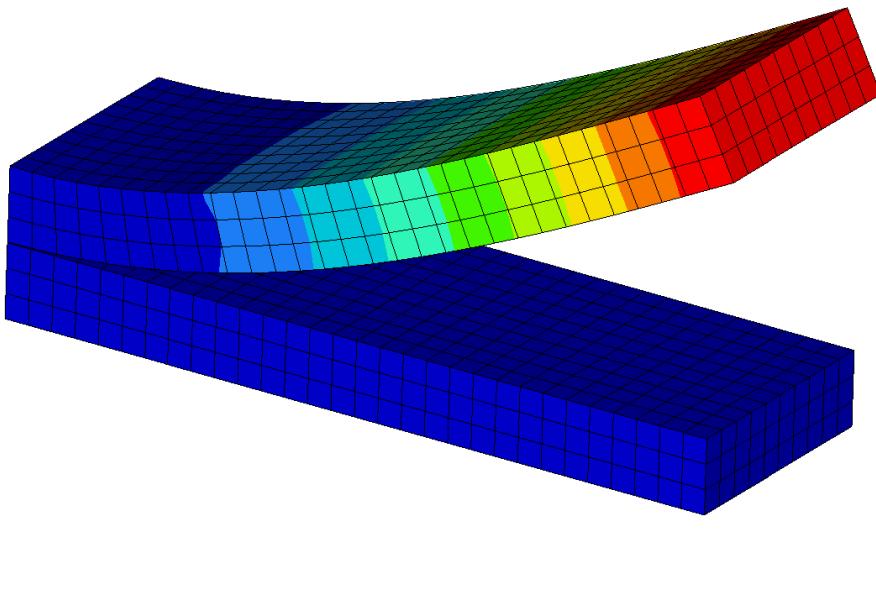
Note the reversed loading (forces pointing upward)

Contour Plot
Displacement(Mag)
Analysis system
■ 2.670E+00
■ 2.373E+00
■ 2.077E+00
■ 1.780E+00
■ 1.483E+00
■ 1.187E+00
■ 8.900E-01
■ 5.933E-01
■ 2.967E-01
■ 0.000E+00
■ No Result
Max = 2.670E+00
Grids 2756
Min = 0.000E+00
Grids 11



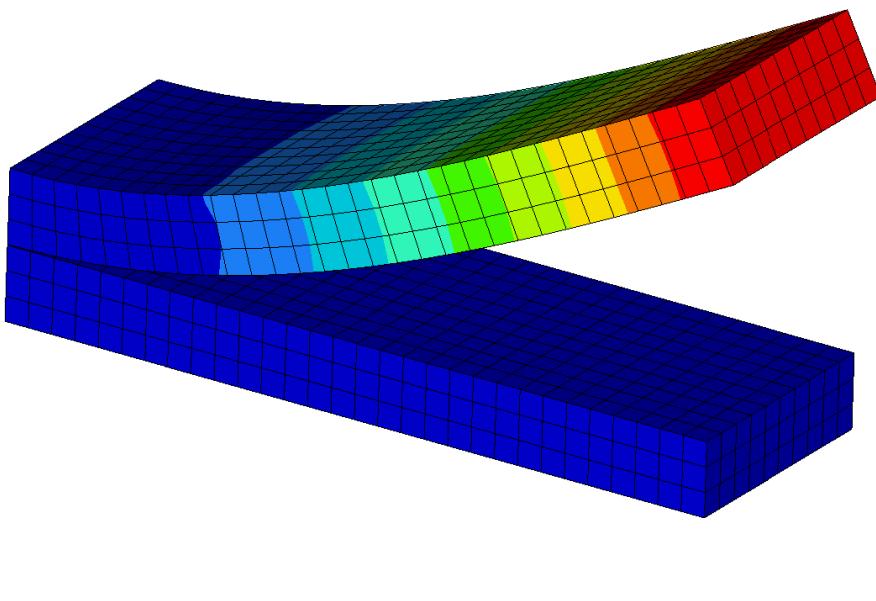
Interface of Type = FREEZE (N2S); displacements are scaled by factor 5. The initial contact gap of 0.25 mm remains constant.

Contour Plot
Displacement(Mag)
Analysis system
-2.063E+01
-1.834E+01
-1.605E+01
-1.376E+01
-1.146E+01
-9.171E+00
-6.878E+00
-4.585E+00
-2.293E+00
0.000E+00
■ Nonlinear stiff
Max = 2.063E+01
Grids 2756
Min = 0.000E+00
Grids 11



Interface of type = STICK (N2S); displacements are scaled by factor 5

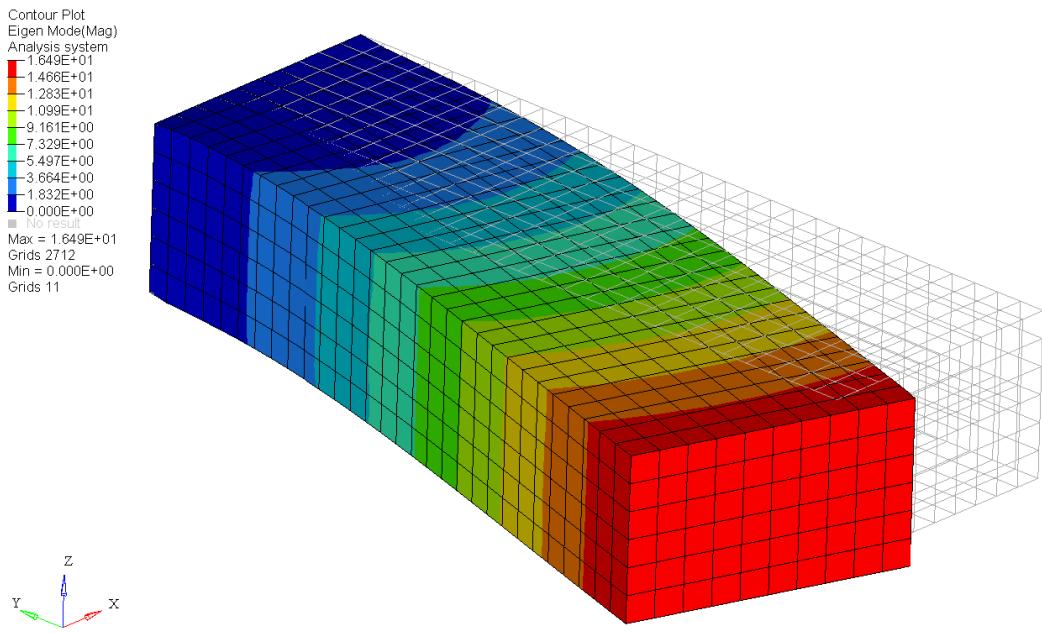
Contour Plot
Displacement(Mag)
Analysis system
-2.063E+01
-1.834E+01
-1.605E+01
-1.376E+01
-1.146E+01
-9.171E+00
-6.878E+00
-4.585E+00
-2.293E+00
0.000E+00
■ Nonlinear stiff
Max = 2.063E+01
Grids 2756
Min = 0.000E+00
Grids 11



Interface of type = SLIDE (N2S); displacements are scaled by factor 5

Did You Know?

Using the FREEZE option a Modal Analysis can be carried out as well. Of course resulting in the same results as if the model would consist of a single block / unit.

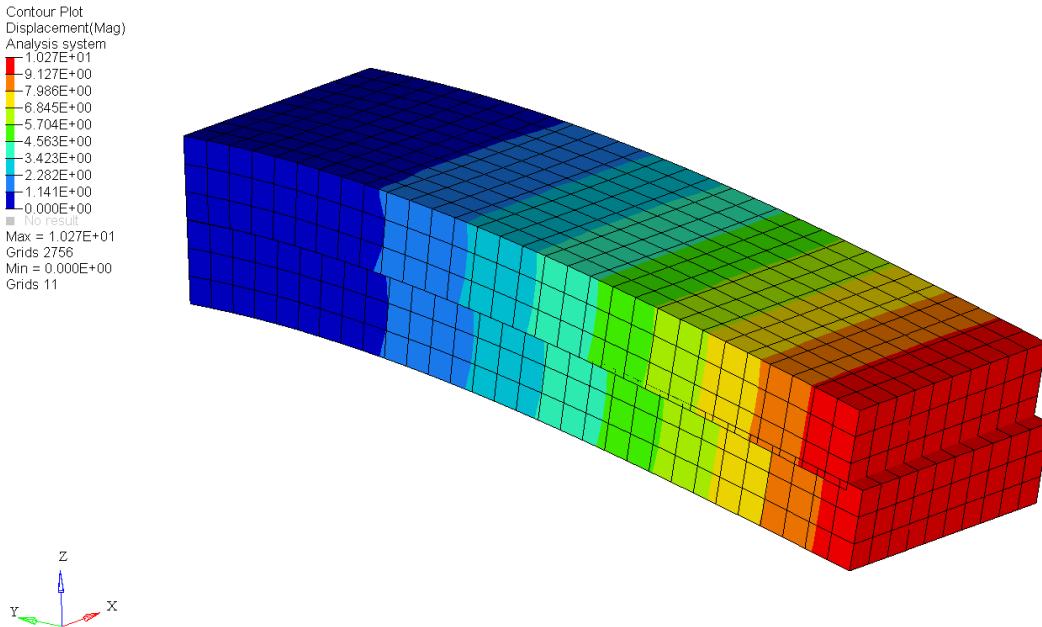


Results of a Modal Analysis, Freeze

A remark about TIE Contact

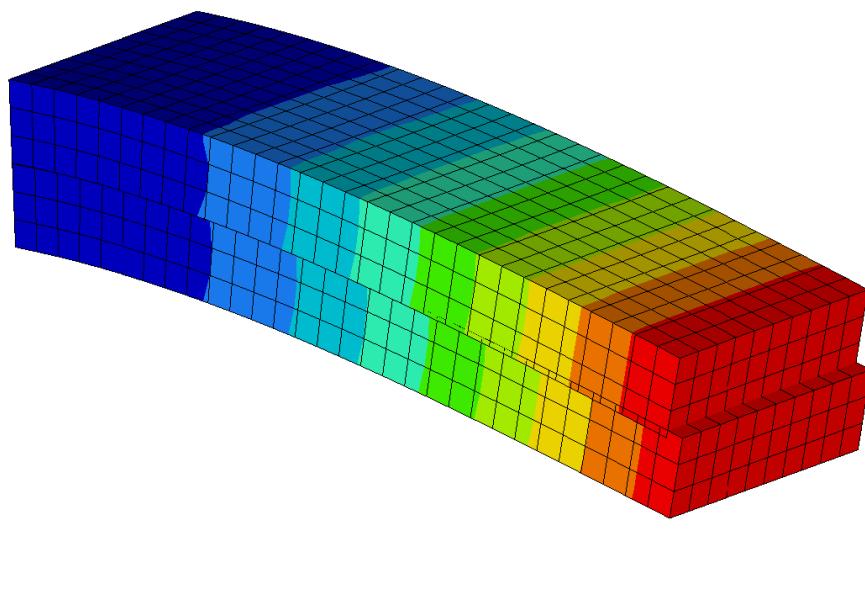
TIE element is created of a same structure as FREEZE CONTACT element. TIE element enforces zero relative motion on the contact surface – the contact gap opening remains fixed at the original value and the sliding distance is forced to be zero. Also, rotations at the slave node are matched to the rotations of the master patch.

Hence, in other words, FREEZE and TIE contact as integrated and available in OptiStruct 13.0 are just the same! Because of simplicity reasons, we therefore recommend to employ the FREEZE contact, following the above described modeling procedure.



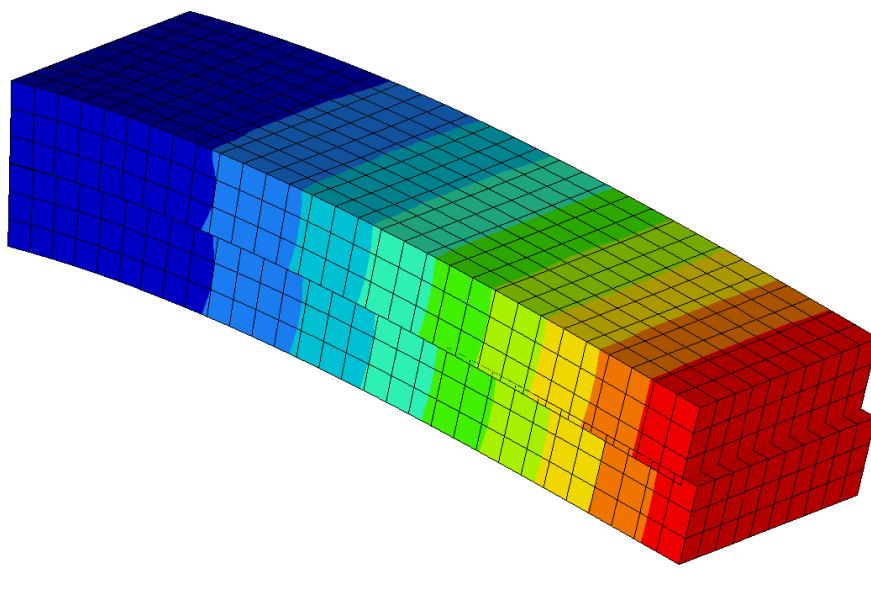
Two blocks with friction MU1=MU2=0.3 (N2S); deformation scaled by factor 5.

Contour Plot
Displacement(Mag)
Analysis system
-1.020E+01
-9.063E+00
-7.930E+00
-6.797E+00
-5.664E+00
-4.532E+00
-3.399E+00
-2.266E+00
-1.133E+00
-0.000E+00
■ No result
Max = 1.020E+01
Grids 2756
Min = 0.000E+00
Grids 11



Contact definition with PCONT, MU1=0.6, MU2=0.4, N2S. Displacements are scaled by factor 5

Contour Plot
Displacement(Mag)
Analysis system
-1.016E+01
-9.031E+00
-7.902E+00
-6.773E+00
-5.645E+00
-4.516E+00
-3.387E+00
-2.258E+00
-1.129E+00
-0.000E+00
■ No result
Max = 1.016E+01
Grids 2756
Min = 0.000E+00
Grids 11



Contact definition with PCONT, MU1=0.6, MU2=0.4, S2S. Displacements are scaled by factor 5

Some Modeling Remedies

- Use incremental loading (NINC = 5 or 10) – friction is path-dependent
- Carefully define CONTACT surfaces:
 - only where contact really expected
 - avoid “by property” contacts (wrapping around entire body)
- Slave on the side with finer mesh

- For second order meshes, use contact-friendly solid elements:
 - PARAM,CONTFEL,YES (check Help Documentation for details)

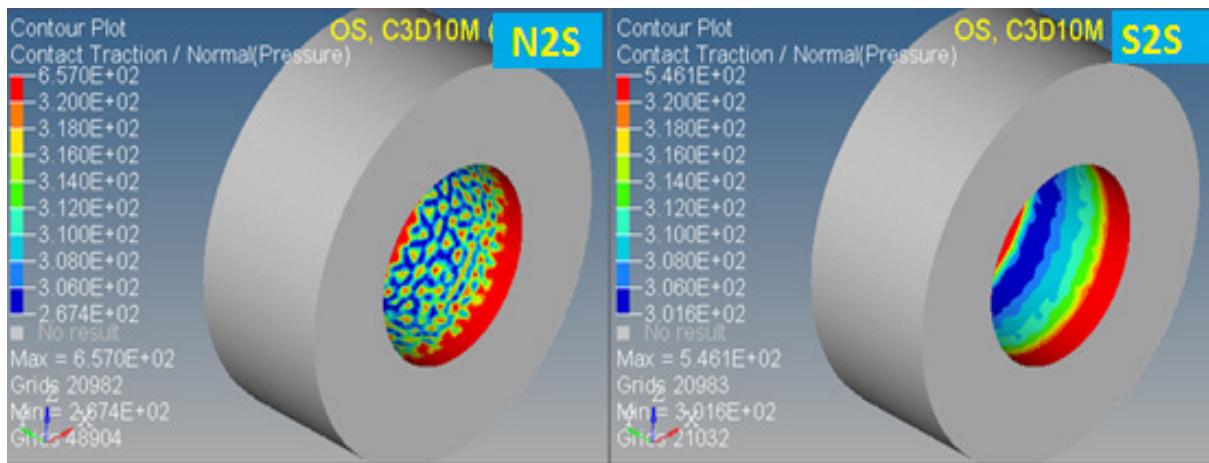
Some Solution Remedies

Surface-to-Surface contact at large deformations

- Can be defined (or switched from N2S) in the CONTACT card

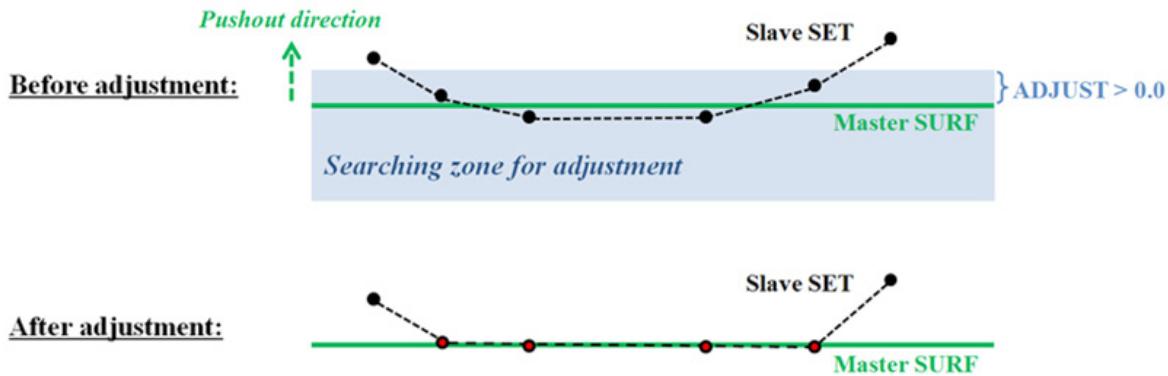
Name	Value
Solver Keyword	CONTACT
Name	blocks
ID	1
Color	■
Include File	[Master Model]
Card Image	CONTACT
Master Entity IDs	top (1)
Slave Entity IDs	bottom (2)
User Comments	Hide In Menu/Export
<input type="checkbox"/> Property Option	Property Id
PID	pcont (3)
MORIENT	
SRCHDIS	
<input type="checkbox"/> Adjust Option	String Value
ADJUST	
DISCRET	S2S
<input type="checkbox"/> Nsm Definition	
Mass	N2S
<input type="checkbox"/> Options	S2S
Element List	0 Elements

- S2S contact formulation is expected to produce better contact pressure for some cases (e.g. relatively flat slave and master surfaces > interpolated)
- Improvement in pressure distribution from S2S contact
 - most pronounced on mis-matching meshes, requires reasonably flat surfaces (no sharp edges)



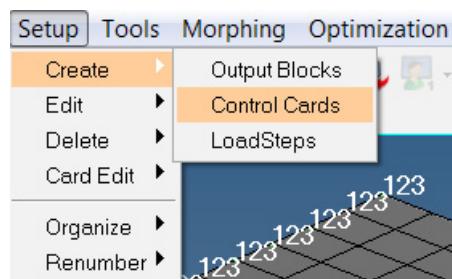
- CONTACT improvements – ADJUST
 - Designed to “smooth out” imperfection of mesh so as to create uniform initial gap
 - Various options of how deep/far the adjust reaches

Name	Value
Solver Keyword	CONTACT
Name	blocks
ID	1
Color	■
Include File	[Master Model]
Card Image	CONTACT
Master Entity IDs	top (1)
Slave Entity IDs	bottom (2)
User Comments	Hide In Menu/Export
Property Option	Property Id
PID	pcont (3)
MORIENT	
SRCHDIS	
Adjust Option	String Value
ADJUST	DISCRET
Nsm Definition	NO
Mass	ALL
Options	AUTO
Element List	0 Elements



- For slow-converging large contact problems – increase MAXITER (50, 100) in the NLPARM Collector
- new default PARAM,EXPERTNL,AUTO will do it automatically. AUTO activates a “light” version of the expert system, which is designed to facilitate converging nonlinear process in reasonably close to minimum number of iterations. In particular, this version may adjust the time step, including increasing the time step beyond that prescribed on the NLPARM card.

Can be activated through



OUTFILE	PROPERTY	SHAPE	
OUTPUT	RESPRINT	SHRES	
P2G	RESTART	SUBTITLE	
PARAM	RESULTS	SWLDPRM	
PFGRID	SCREEN	SYSSETTING	
PFMODE	SENSITIVITY	THICKNESS	
PPANEL	SENSOUT	TITLE	

delete
disable
enable

next
prev
return

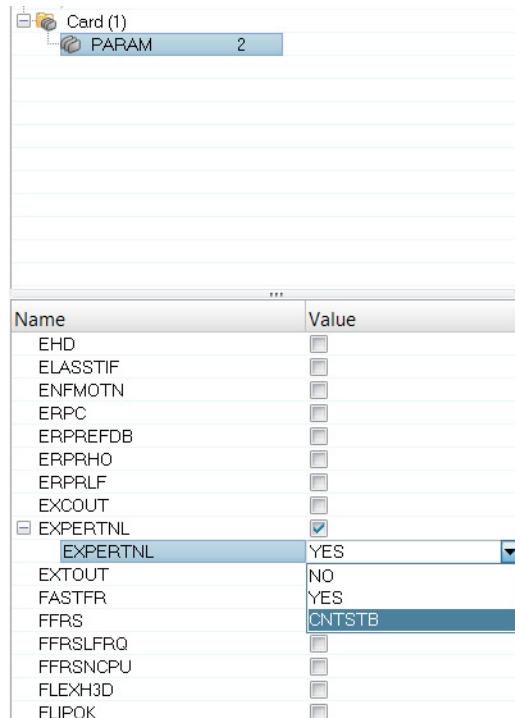
EXPERTNL	
PARAM , EXPERTNL ,	YES

◀ □ ERPRREFDB
□ ERPRHO
□ ERPRLF
□ EXCOUT
✓ EXPERTNL
□ EXTOUT
□ FASTFR

reject
default

abort
return

- Try stabilized expert system: PARAM, EXPERTNL, CNTSTB. Especially on models that “blow up” due to insufficient support.
- CNTSTB additionally introduces temporary stabilization on contact interfaces (CONTACT or GAP(G) elements) that may improve nonlinear convergence, especially in cases where individual parts lack full support and are supported only by contact. The stabilization is applied only during incremental loading and is not present in the final solution for the respective nonlinear subcase.



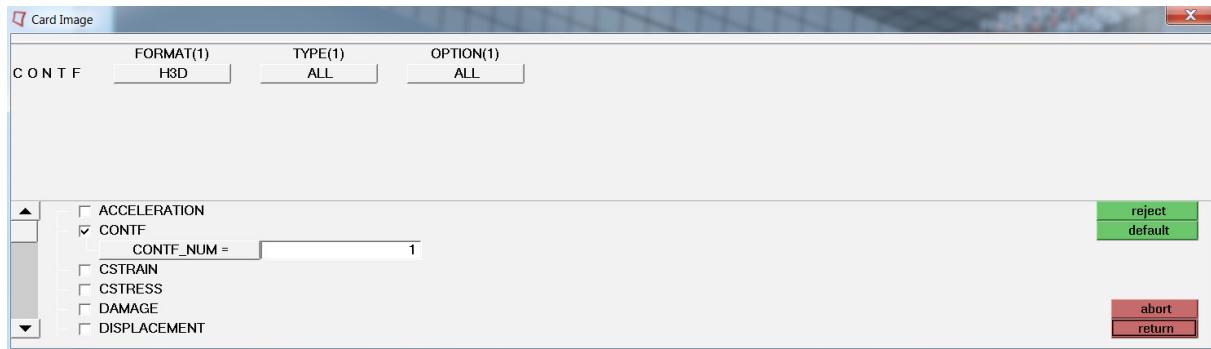
- Verify whether FREEZE works
- All contacts sliding? Friction may be too low
- In addition you may consider consider the option:
 - STIFF=SOFT on PCONT

Option STIFF=AUTO determines the value of normal stiffness for each contact element using the stiffness of surrounding elements. Additional options SOFT and HARD create respectively softer or harder penalties. SOFT can be used in cases of convergence difficulties and HARD can be used if undesirable penetration is detected in the solution. A negative value for STIFF indicates that a stiffness scaling factor equal to $|Real < 0.0|$ is defined. This scaling is applied on the stiffness value via STIFF = AUTO.

Postprocessing Remedies

Request More Contact Results

To request additional results data, activate the CONTF Card in Control Cards □ Global Output (see below).



Note: The CONTF entry is only applicable in nonlinear analysis subcases that are identified by the presence of an [NLPARM](#) subcase entry.

In the legend displayed in HyperView you will see the following notifications:

Open/Closed status is represented by 0.0 for Open and 1.0 for Closed. On graphical display, intermediate values may appear due to transition from open to closed across individual elements.

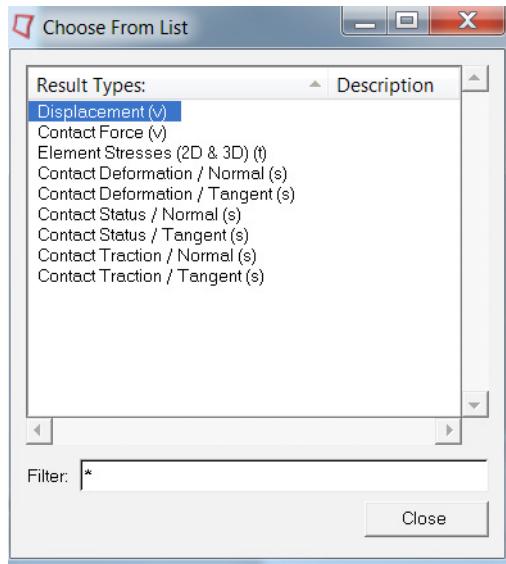
Slip/Stick Status is represented by 0.0 for Open, 1.0 for Slip and 2.0 for Stick. On graphical display, intermediate values may appear due to transition of status across individual elements.

Sliding Distance represents total sliding distance accumulated while the surfaces are in contact. This may be different than just the difference in displacements between the starting and final position.

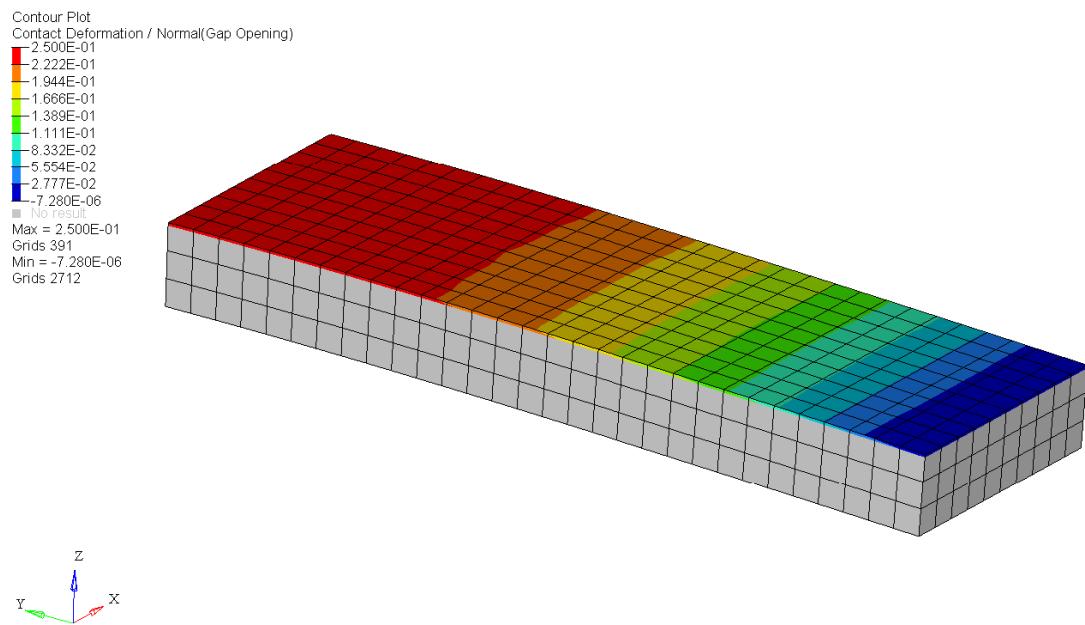
In the Browser the following information will be depicted:

Name	Value
Solver Keyword	CONTF
Include File	[Master Model]
Status	<input checked="" type="checkbox"/>
ACCELERATION	<input checked="" type="checkbox"/>
CONTF	<input checked="" type="checkbox"/>
CONTF_NUM =	1
GLOBAL_OUTPUT_REQUEST	<input checked="" type="checkbox"/>
FORMAT	H3D
TYPE	ALL
OPTION	ALL

In the *.h3d file the additional contact results will be available:



For instance, in the image below the gap opening (initial opening 0.25 mm) is depicted:

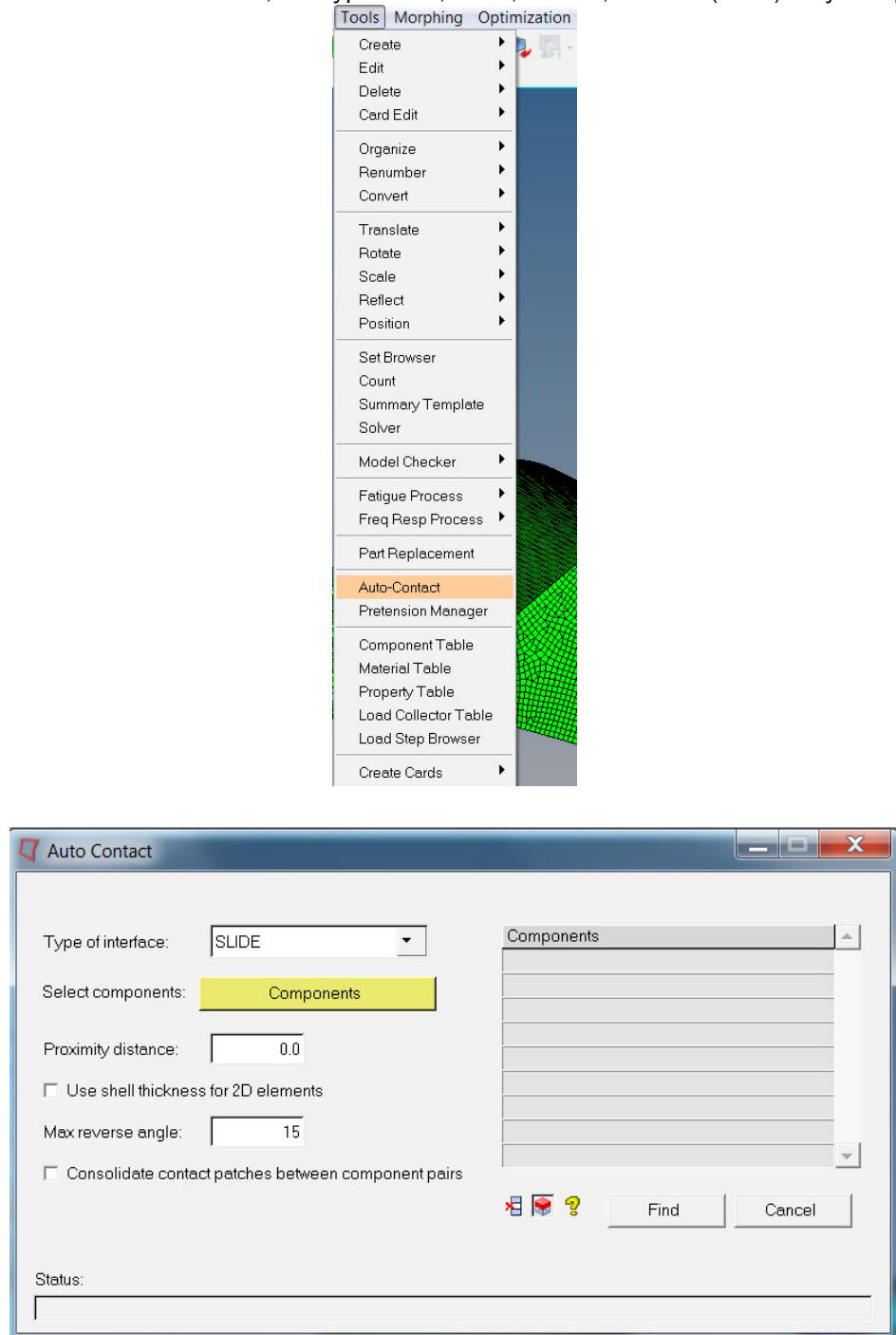


19.13 Auto-Contact Manager

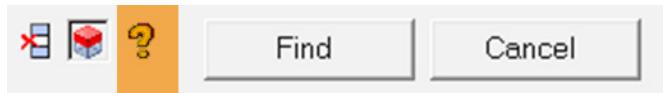
A very different way on how to create contacts – especially for large models with many different contacts – is to employ the so called Auto-Contact Manager.

The Auto-Contact Manager automatically searches for possible contacts across selected components based on proximity distance. For components within the proximity distance interfaces of type CONTACT are created.

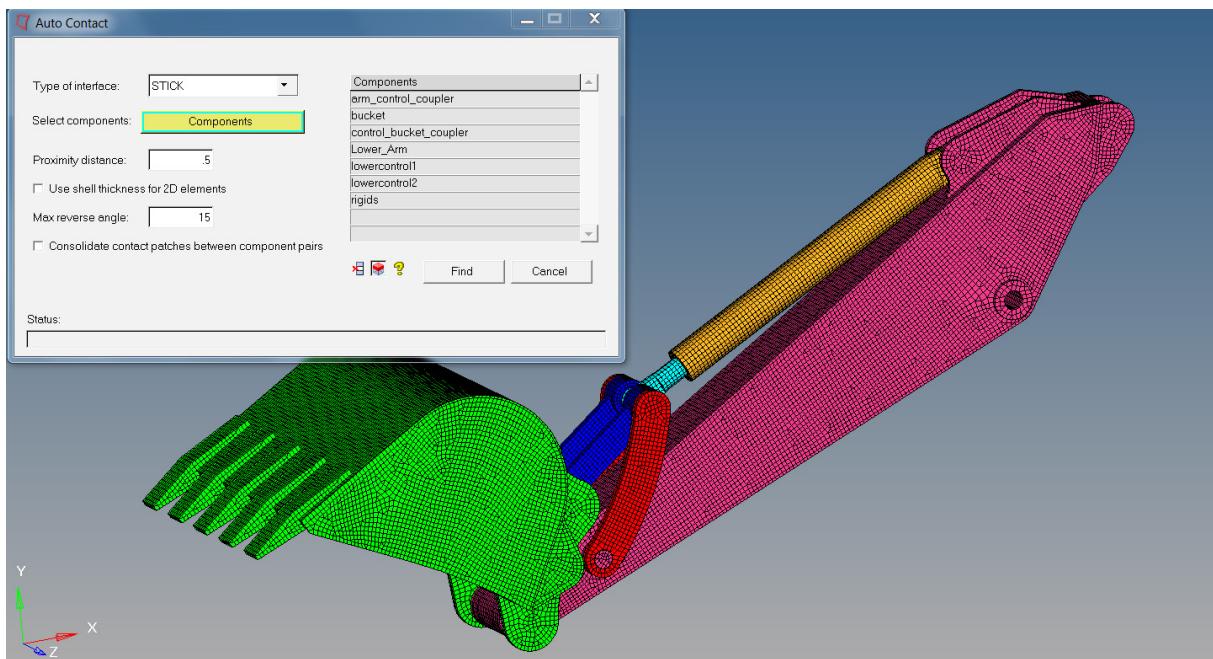
As in the manual process described above, the type Slide, Stick, Freeze, Friction (MU1) may be specified.



Note: A detailed description about e.g. "Use shell thickness for 2D elements" and other options can be accessed by clicking on the "?" symbol



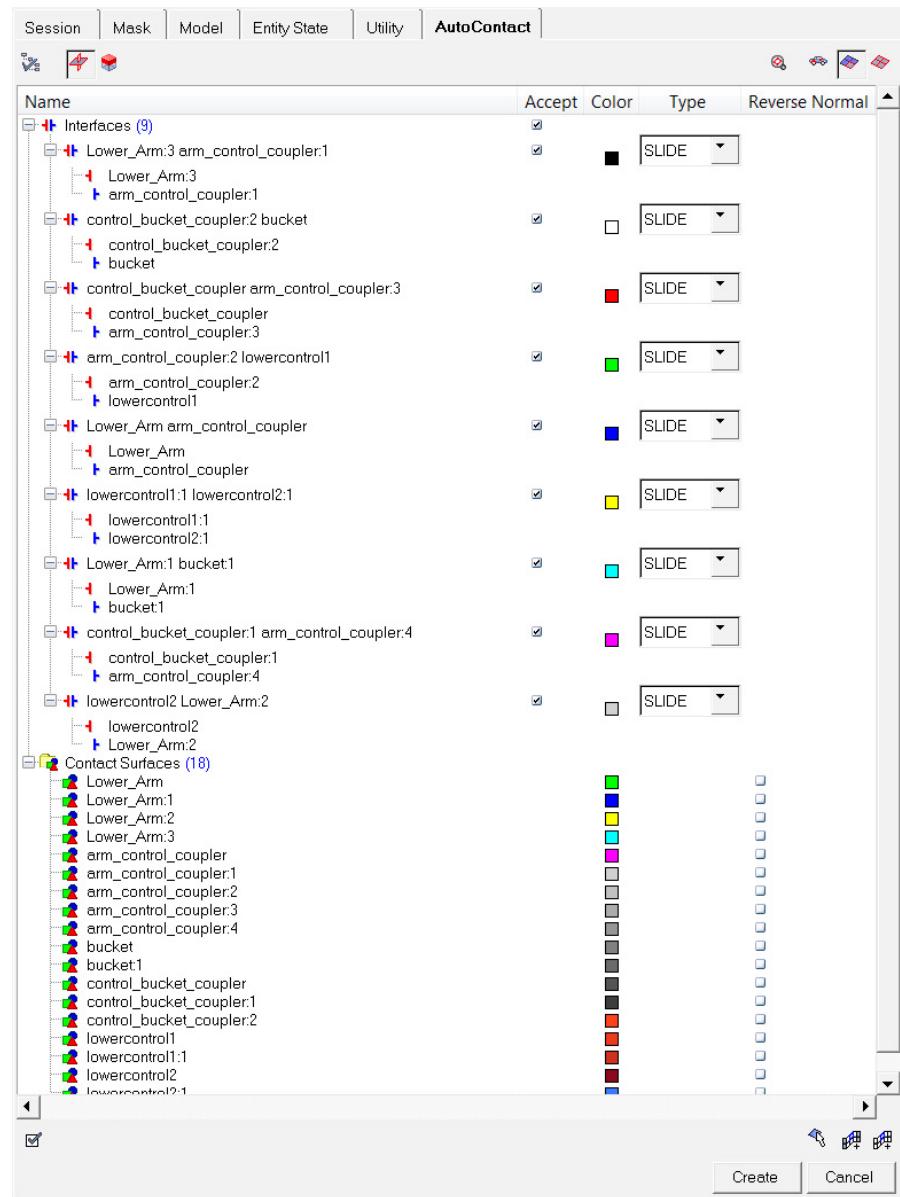
After selecting the components of interest (e.g. all, displayed), insert the proximity distance and click Find to start the search for likely contacts in the model.



Likely contacts will then be listed in the Auto-Contact Browser. The Auto-Contact Browser allows to review and to edit the Contact Surfaces and Interfaces.

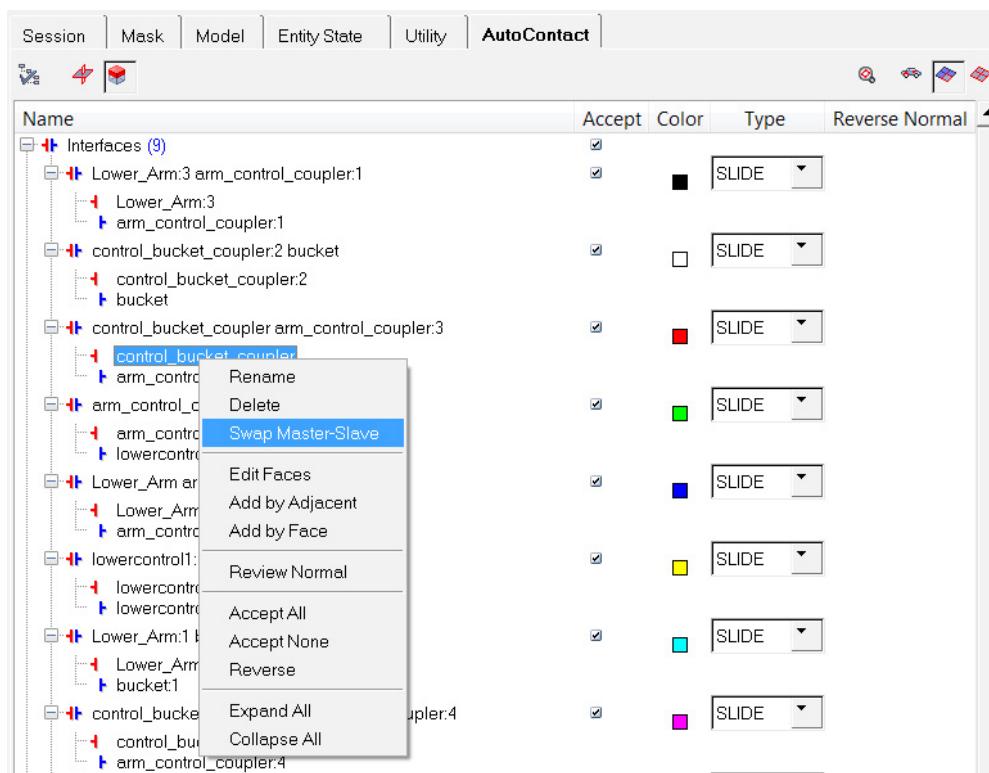
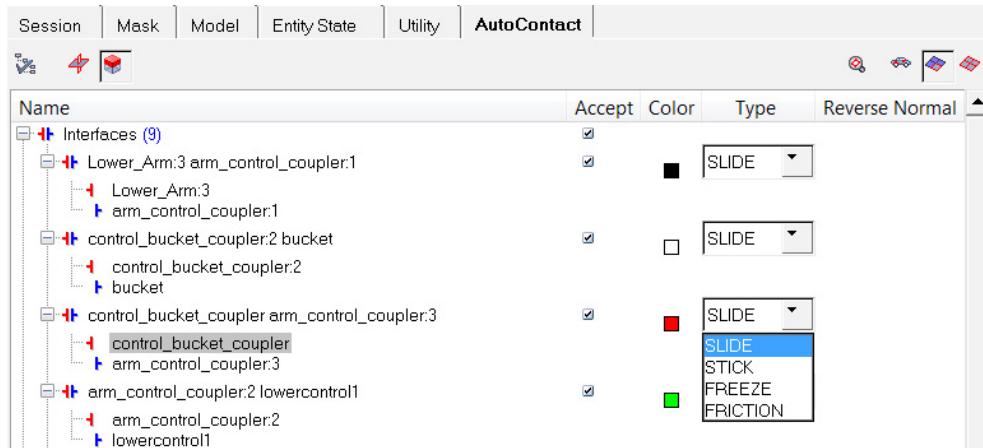
Remarks

During the Auto Contact process, temporary components may be created for parts containing 3D elements. These temporary components will be named using a preceding ^ symbol, and will automatically contain extracted element faces needed for the contact creation process. Auto Contact will cleanup and remove these temporary components when you finish or cancel the process. If you decide to export the model before accepting or canceling the process, HyperMesh will always exclude these temporary components from export.

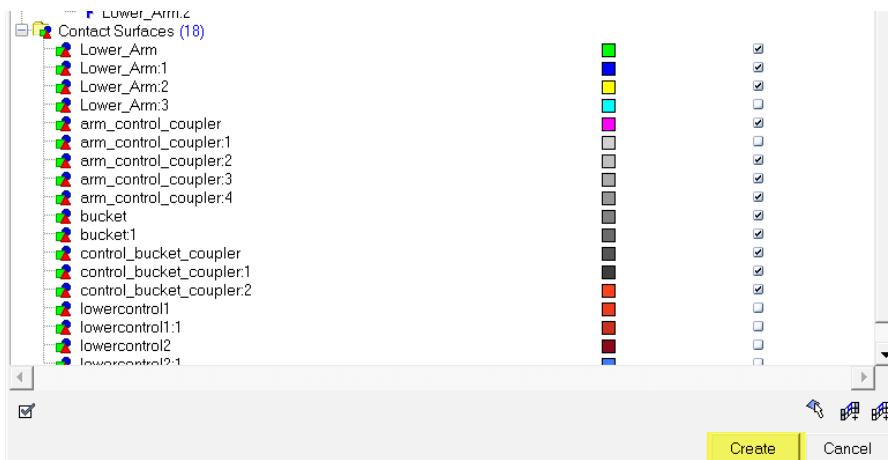


In detail you can pursue the following tasks:

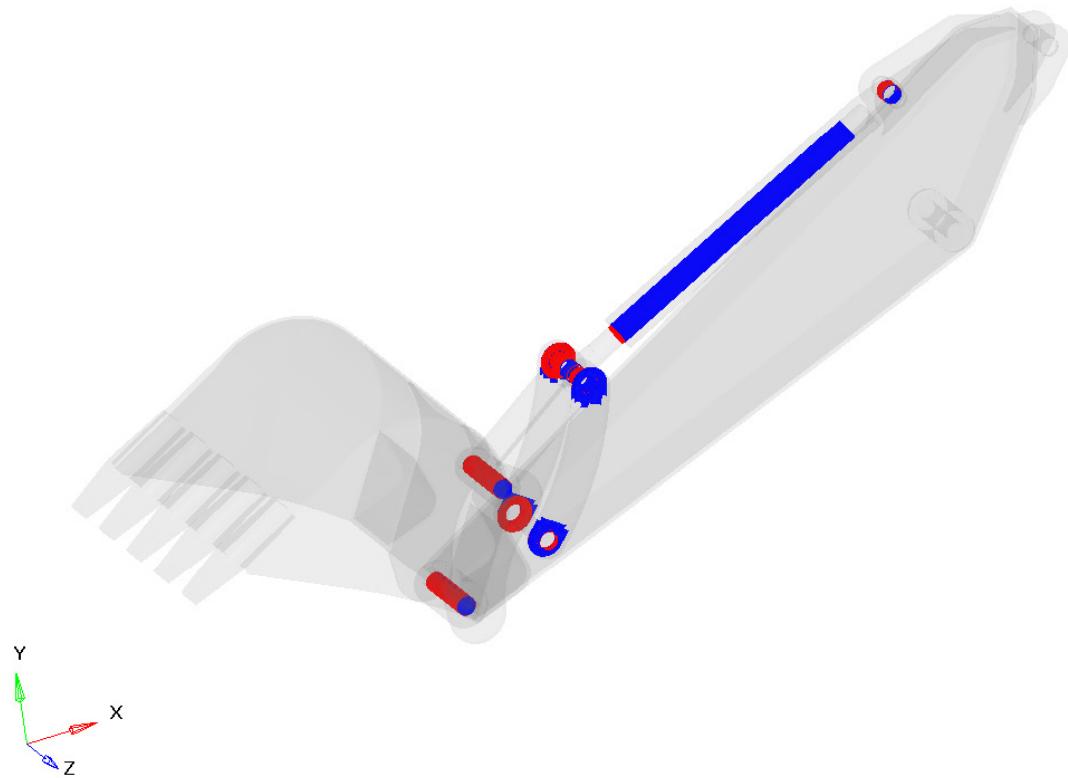
- Contact TYPE editing.
- Master/Slave swapping.
- Contact surface quick editing:
- Auto-add elems by adjacent
- Auto-add elems by face
- Add elems interactively
- 2D element normal review
- For 3D elements, contact surfaces are automatically oriented outwards.
- Contact Surface normal reverse option



Eventually, the contact elements are created after you clicked on „Create“ (in the lower right corner of the browser).



To review the interfaces, especially to see their location the following viewing mode is helpful

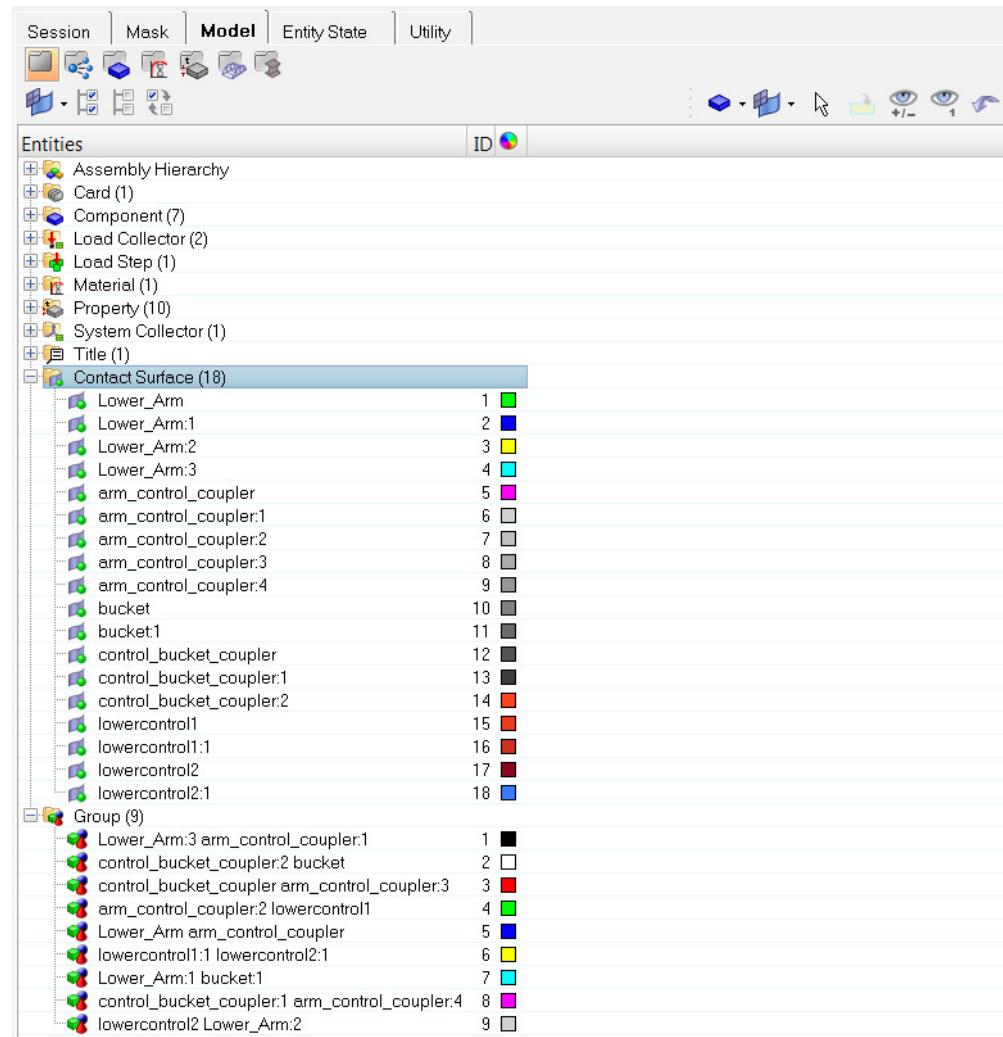


which can be activated through



Review elements & Display components with elements

The Auto-Contact Browser disappears right after the contacts are created, i.e. now you see Contact Surfaces and Groups (=interfaces) being listed in the Model Browser.



This process is pretty straight forward and pretty fast. With the overview provided in this chapter you should be ready to get started with your first Contact challenge.

Success with your first Contact Model.

The following article was kindly provided by

Professor R. Steinbuch, Department of Engineering, Reutlingen University, Alteburgstraße 150, D-72762 Reutlingen, Germany;
e-mail: rolf.steinbuch@reutlingen-university.de

19.14 Contact Modeling in Structural Simulation – Approaches, Problems and Chances

Abstract

Nearly all structural problems deal to some extend with interacting bodies [1]. In consequence, contact between these bodies is one of the most often occurring problems in modelling. On the other hand, many contacts are either neglected in modelling or assumed to be represented by appropriate boundary conditions. Today most of the simulation codes used in structural mechanics allow to model contact. In consequence there is a need to qualify more users to apply contact models in everyday simulation practice. Modelling contact implies to consider the appropriate setting of the available contact parameters. Using the standard values proposed by the codes or unqualified selection of parameter combinations may produce simulation results which are either poor or even misleading. To stimulate the discussion about contact modelling some of the aspects to be considered and their non-unique consequences are discussed.

Choosing qualified element types surely will influence significantly the simulation velocity and quality. Adapting meshes to the current state of the problem may improve speed and quality essentially, but mapping the analysis results from old meshes to the new ones may cause loss of local quality. Reduced integration helps to speed up the computing time and avoid numerical stiffness, but the stress results are less qualified. Artificial stiffness introduced to prevent misleading effects like hourglasses produces potential sources of numerical errors. The ways to detect and release contact in time and space influence significantly the course of the analysis. Simple approaches yield fast responses at the price of incorrect penetrations and incomplete definitions. Qualified algorithms may converge only after large numbers of recycles and time step reductions, so acceptable response times may not occur. Friction models suffer from the uncertainty of data and the large scatter of their input values, especially in transitions of stick to slip, high speed sliding and lubrication. The decision whether to use explicit or implicit time integration is not easy to be made in many cases.

A qualified discussion of the interaction of the parameters in contact analysis will certainly not yield a simple scheme applicable in all cases required to be considered in structural simulation. Users will be faced with problems where the ideas they have been applying successfully in many studies fail, do not converge in appropriate time or even come up with misleading results without indicating the potential danger. So the most important parameter in engineering contact analysis is the experience of the people running the jobs. Some application examples underline the problems mentioned and sketch solution proposals for some parameter configurations. Critical regions are defined, workarounds proposed and experiences presented. Small parameter changes may cause large changes in system response. In consequence for contact problems robustness studies, indicating the sensitivity of the physical and numerical parameters should be performed. In any case the qualification and care of the analyst applying contact methods in engineering remain the most important tools to solve contact problems successfully and to learn about the interaction of structures.

Contact In Structural Problems

Nearly all coherent structures like metal parts, stones, plants or furniture, to mention some, interact with their surrounding via surface contact. A table is standing on the floor, the ends of its legs contacting the surface of the floor of the room. The only exceptions of bodies not in structural contact with some neighbours are structural units flying through the air or the empty space like meteorites. So contact and the necessity to handle contact is not an exotic task, but one of the basic questions in structural mechanics. But as contact problems are often related to nonlinear phenomena many engineering approaches try to avoid contact modelling by the use of linear surrogates.

- **Boundary Conditions Or Contact?**

In mechanics the idea of free body models, replacing all interaction with the contacting surrounding by application of corresponding forces or moments is crucial to start the studies of static problems. Introducing contact models at that early state of education

would increase the confusion in the novices' heads, who are struggling hard enough to enter the world of mechanics. So models like the one shown in Figure 1a are supposed to represent real support loading situations indicated in Figure 1b. Effects like local notches or sharp corners between the interacting bodies are ignored and believed to be of none or little influence. But when dealing with real components and their failure modes, we realise, that a good part of the problems are caused by these local disturbances we neglected before.

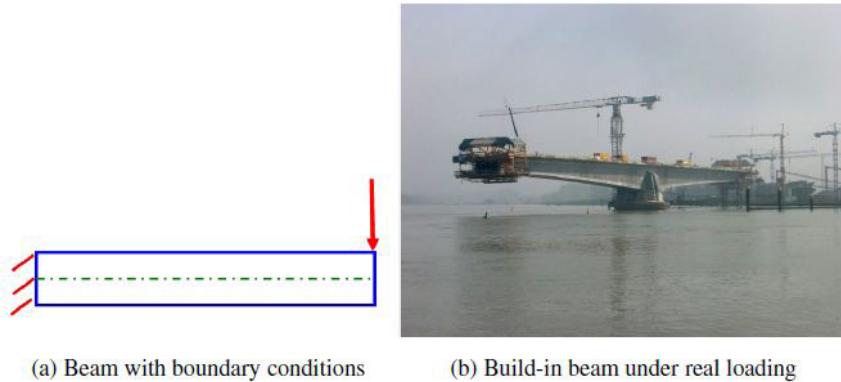
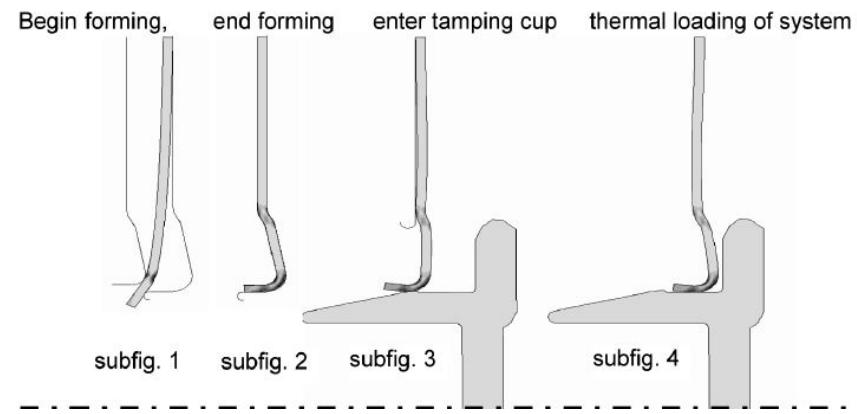


Fig. 1 Idealised and real loading situation of beams.

- **Contact As Part Of Manufacturing Processes**

Contact, being present as a static boundary condition in most structural systems, is additionally defining some of the processes in mechanics. All the shaping of structural but deformable material including unwanted forming like crash is specified by some contact history. Segments of parts surfaces come into contact, the acting forces cause deformation of the surrounding material [2]. New contacts are possible due to the fact that material preventing the contact has been removed by the contact process. Fig. 2.a demonstrates some steps in an axisymmetric deep drawing process, using different tools and finally covering the newly manufactured can by a seal.



During two steps of forming, the shape of the top of the can is manufactured (subfig. 1 and 2). Then the tampering head is introduced (subfig. 3). Finally some thermal load steps caused by temperature changes of the whole system are applied (subfig. 4). The analysis aimed to check if the sealing by the tampering head would be sufficient during the lifetime of the assembly. The contact study performed checks whether the manufacturing is possible, if problems may occur, e.g. large strains may indicate the failure of the sheet metal and other related quantities like the spring-back after the removal of the die. In addition including the deep drawing into the system's analysis helps to come up with a better evaluation of the structure.

Figure 2b plots the sealing forces between the can and the tamping due to temperature changes. Obviously the study not including the deep drawing steps (w/o manufacturing) predicts a failure of the seal during the thermal loading (radial pressure = 0) while the more elaborate simulation (with manufacture) predicts a sufficient tightness (radial pressure > 0).

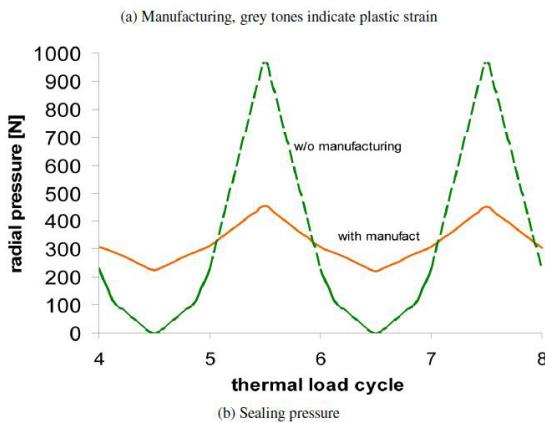


Fig. 2 Some steps of a deep drawing of the top of a can (axisymmetric) and sealing.

- Numerical Contact Studies

If two bodies come into contact there is a small region of first encounter, which will be increasing, if the bodies are continuing to move in the closing direction. Perhaps some of the early contact regions are released from contacting, while others support the load to be transmitted via the contact areas. A continuous process of establishing and releasing contacts will evolve. When studying more complex problems of continuum mechanics it is well known that there are no analytical or classical solutions available as soon as the problems exceed a certain level of simplicity. Discrete numerical strategies like FEM or related approaches are used to analyse these nontrivial problems. But these discrete systems have the disadvantage that their discrete schemes including discrete discovery and release of contact will fail to reproduce the soft initiation of continuous contact. Figure 3 compares the smooth rolling of a circular wheel with the inevitable rumbling of a discrete representation of this wheel.

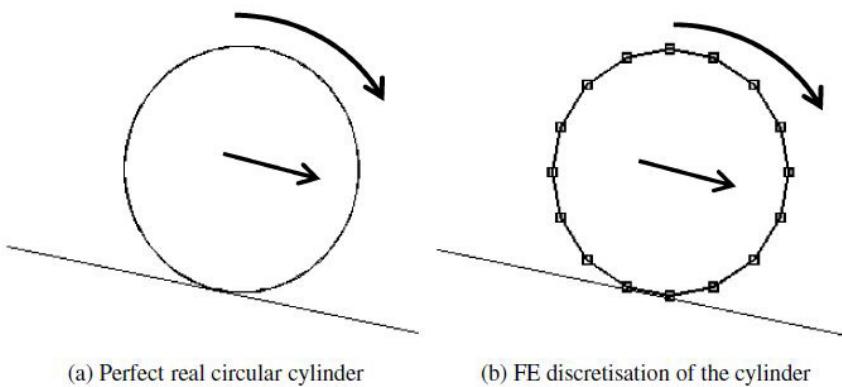


Fig. 3 Rolling of a cylinder and a FE-model of the cylinder on an inclined plane.

While Figure 4a depicts a pipe bending study, Figure 4b presents the local contact forces during this pipe bending study at a given time step. There are sharp changes in the contact state and forces at the Finite Element nodes, which is quite the opposite of the real smooth process in time.

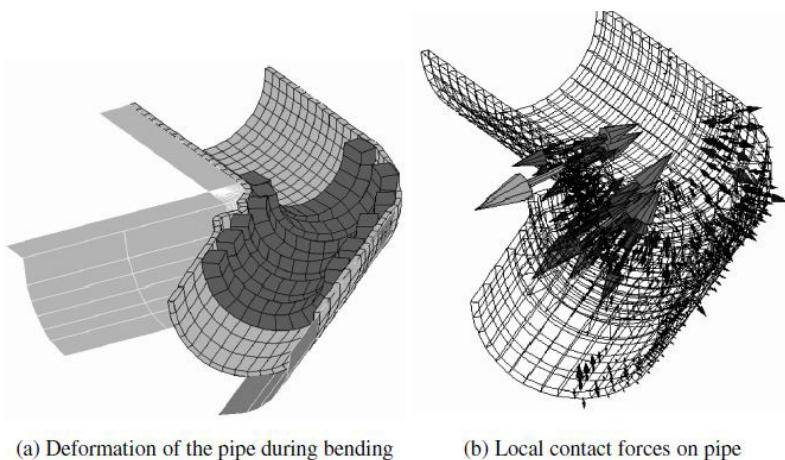


Fig. 4 Pipe bending study.

Terms And Definitions

Having mentioned some of the basic terms of contact modelling the need to clarify terms used in contact studies arises.

- **Real Contact**

The following short definitions for contact between real parts hold and are generally accepted (see Figure 5a):

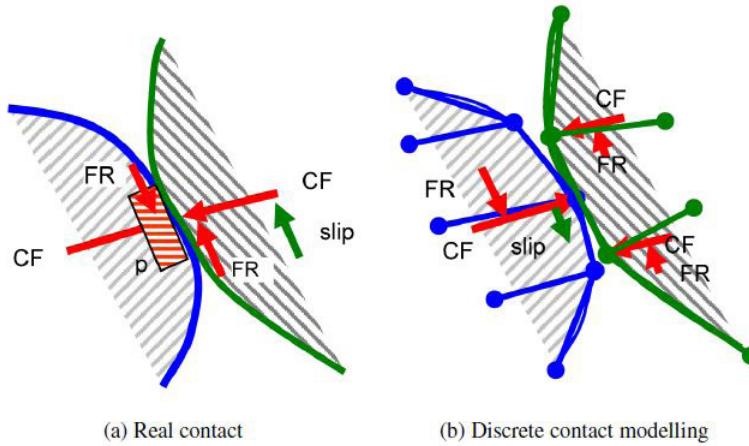


Fig. 5 Main contact parameters.

Contact: Two bodies meet, are in structural interaction, the contact normal force is positive.

Release: The contact ends, no more structural interaction, the contact normal force is zero.

Contact force: Force normal to the contact area acting between the contacting bodies (CF).

Contact pressure: Contact force/contact area (p).

Friction: Force transverse to contact closing direction, opposing the sliding tendency (FR).

Stick: No relative transverse movement, often due to friction.

Slip: Sliding along contact surface overpowering the sticking friction forces.

- **Numerical Contact**

The same terms used for real contact may be used for the discrete numerical modelling of contact. We are taking into account that no longer areas are interacting, but discrete nodes or points are checked, if they are penetrating opposite surfaces. All contact properties, especially normal and friction forces, stick or slip responses have to be transferred to the discrete nodes like indicated in Figure 5b.

- **Parameters Used In Contact Studies**

Going deeper into the problem of modelling, the process so simple to describe experimentally exhibits a large number of parameters to be covered. Table 1 tries to identify some of them, including the most commonly ways used to handle them.

Table 1 Some of the major contact modelling parameters.

Parameter	Values		
elements	linear	quadratic	higher order
adaptivity	refine	coarsen	remesh
element integration	reduced	full	–
deformation	elasto-plastic	rigid-plastic	–
contact type	master-slave	both partners	contact range
contact partners	flexible vs. stiff	both flexible	–
contact modelling	penalty	Lagrange multipliers	merging
shell contact	include thickness	neglect thickness	1 or 2 sided
corner handling	one side only	slip around	–
catching	range	after penetration	–
release	immediately	slowly	–
friction	Coulomb	shear	glue
friction definition	velocity	displacement	–
friction onset	stick-slip	continuous	–
time integration	explicit	implicit	–
time modelling	static	dynamic	large masses

Neither the completeness of the parameters list nor the proposed handling must be accepted by other contact modelling engineers or systems. Nevertheless Table 1 gives some idea that contact modelling is not a simple task. Furthermore it tries to motivate potential users not to trust their codes without counterchecking the default values proposed by the different simulation systems. The users should analyse their problems, compare the specific requirements with the possibilities the codes offer and search for reasonable solutions. There is no commonly accepted agreement about the preferences to be made, the parameter combinations to be used or unwanted interactions of parameter choices to be avoided. In many cases extensive studies have to be performed before a reliable contact analysis may converge to a satisfactory result.

Two examples may enlighten this warning. Linear elements are simple to be used, but require small contact edges, while quadratic elements may yield faster results using essentially smaller node and element numbers. On the other hand, the local forces of quadratic elements are different at the corner and mid-side nodes. Larger local forces occur at the midside nodes than at the edge nodes. As a consequence the forces transmitted locally between contacting structures may not be correctly placed. Explicit time integration is known to produce fast results at relatively small time and storage requirement, while implicit time integration needs large storage and lots of time for the matrix inversion.

Some Problems And Some Proposals

Some of the contact parameters and related problems should be looked at, to come up with an understanding of the necessity of sensible handling contact simulation.

- **Element Types And Contact Definition**

Contact is (see Figure 5) the interaction between two bodies. The result of the modelled contact should not depend too much on the modelling approach. Figure 6 however indicates that local effects may vary significantly due to the element size and element type used. Only for very fine meshes, the desired symmetric and element type independent result occurs. This leads to the proposal to use very fine meshes or sensitive adaptive algorithms

- **Contact State**

The definition of the contact closing and the removing of penetrations may be done by many strategies, Lagrange Multipliers and Penalty Methods being the most popular ones. Some codes propose locally conforming meshes, merging the respective nodes

or degrees of freedom. Others apply forces instead of displacement constraints to remove overlapping of the contacting parts.

Even the definition when contact happens is not unique, master-slave and symmetric checking strategies show acceptable results, but sometimes yield different contact response.

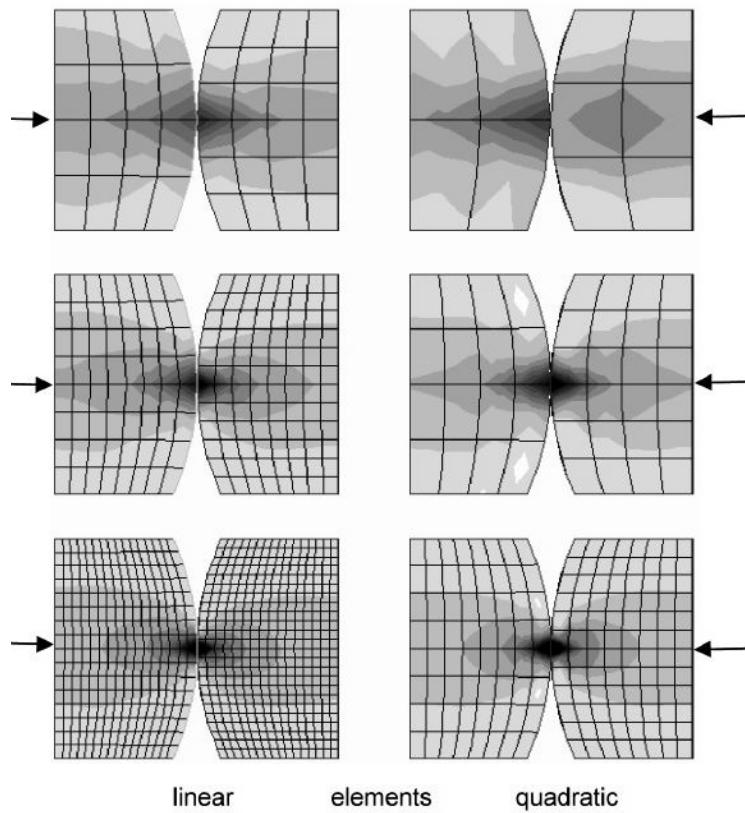


Fig. 6 Contact convergence, grey tones indicate contact normal stresses.

- **Friction**

Friction, the force acting normal to the contact closure direction and parallel to the contacting surfaces is one of the most difficult and little understood phenomena in contact studies. In the microscopic scale local structures interact, build bridges, hooks, joints, are removed by sliding of the surfaces and rebuild at a new position.

Molinari [3] presents impressive studies of local interaction yielding local forces and the resulting stresses. Simulation codes generally use coarse simplification of the friction effects by introducing friction coefficients, sometimes dependent on the local pressure and sliding velocity. Nevertheless evenmore qualified ideas are hardly able to represent the friction of real surfaces. Figure 7 demonstrates for two simple 2D-surfaces, how the local friction depends on the meshing parameters, even the refinement of the meshes produces no tendency of convergence.

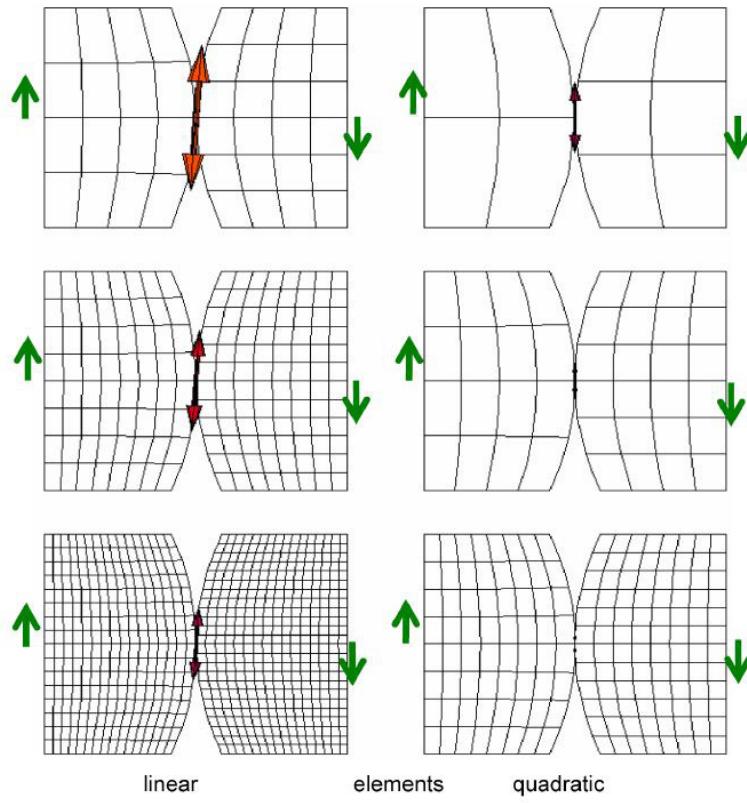


Fig. 7 Friction as function of element type and size, outside arrows indicate transverse displacement, inner arrows indicate local friction force.

- **Time History Integration**

Contact is the result of the motion of different bodies, which initially had some distance or little common surface. During the contact process the area of interaction, the contact surfaces vary as well as the local and global forces. Regions being initially free are detected by free surfaces of the other contact partners, contacting regions are released or at least carry a reduced load. There are some promising approaches to use the final relative position of the bodies for the analysis [4, 5]. In most cases however it is a good idea to start with free bodies. Then one defines a motion, bringing into contact the bodies like indicated in Figure 8. From the initial independent state contact surfaces are evolving, showing a time dependent local response, again different for linear and quadratic elements.

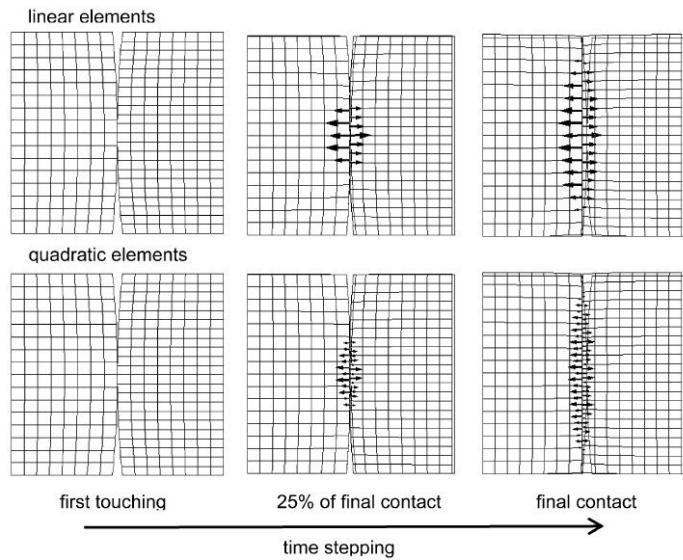


Fig. 8 Time dependent contact surfaces and local forces.

- **Time Integration Schemes**

There are many discussions about the procedures of time integration. Since the “early days” scientists discuss whether to use explicit or implicit time integration schemes. From the classical point of view there is no doubt, that the implicit solution

$$K_n \mathbf{u}_n = \mathbf{F}_n \quad (1)$$

or its dynamic version

$$M_n \ddot{\mathbf{u}}_n + C_n \dot{\mathbf{u}}_n + K_n \mathbf{u}_n = \mathbf{F}_n \quad (2)$$

where the index n indicates the new time step, should be preferred, as it implies the equilibrium of forces at the new time step. In consequence relatively large time steps may still yield sufficient quality of the numerical prediction of the time history. Unfortunately for both static and dynamic problems the need to solve large linear equation systems requires large computation time and storage requirements. Especially in the case of metal forming or crash one needs to recycle the analysis at every time step until satisfactory convergence in terms of equilibrium of internal and external forces in equations (1) or (2) is achieved. This makes implicit time integration an expensive, in many cases an unacceptable tool. So the upcoming of explicit integration schemes marked an essential step towards the feasibility of large contact studies. Assuming that in small time increments, things will not change essentially, the equation to be solved is

$$K_o \Delta \mathbf{u} = \Delta \mathbf{F} \quad (3)$$

respectively,

$$M_o \ddot{\mathbf{u}}_o + C_o \dot{\mathbf{u}}_o + K_o \mathbf{u}_o = \mathbf{F}_o \quad (4)$$

the index o now indicating the old stiffness, mass and damping matrix, displacement and force. Substituting the finite difference approximations for the time derivates and using diagonal mass and damping matrices, equation (4) yields

$$M_o \frac{\mathbf{u}_n - 2\mathbf{u}_o + \mathbf{u}_{o-1}}{\Delta t^2} + C_o \frac{\mathbf{u}_n - \mathbf{u}_{o-1}}{2\Delta t} + K_o \mathbf{u}_o = \mathbf{F}_o \quad (5)$$

\mathbf{u}_n denoting the new displacement again, \mathbf{u}_{o-1} standing for the past time step's displacement.

From equation (5) we find the new displacement \mathbf{u}_n . Using equations (4) and (5) has the big advantage that the stiffness matrix has never to be assembled or inverted. So the CPU-time and storage requirements to perform one time step are reduced by large factors. On the other hand the time steps have to be reduced essentially to avoid to large an accumulation of errors due to the fact that the equilibrium at the old time step is used to predict the new displacements. The numerical limit to the time step enforces even smaller time increments.

To give some ideas about the impact of the time integration scheme we remember the pipe bending example demonstrated in Figure 4. The CPU-time and the storage requirements are compared in Table 2. Obviously the explicit code has large advantages. But on the other hand, the fact that explicit codes always yield results is not without drawbacks. When implicit codes show little stability or fail to converge, this may indicate that physical problems arise. So the deficit in numerical performance may be an advantage, motivating the qualified analyst to improve his ideas, to check assumptions, to go deeper into the details of the problem posed.

Table 2 Comparison of implicit and explicit resources used for pipe bending.

Integration	Processors	RAM [GByte]	CPU-Time [h]	Rel. velocity
implicit	16	24	8	1
explicit [9]	2	2	2	32

- **Reliability Of Codes**

One of the most dangerous obstacles in simulation of contact is the reliability of the commercial or private codes. Nonlinearity and contact are included in most today modelling systems, even if sometimes doubtful contact options are offered to the user. Unfortunately many of the options and parameters are neither clearly understood nor defined. In addition, the difficulty of contact modelling makes the software provider proposing many different parameter schemes. Even high level quality assurance does

not inhibit erroneous codes as indicated in Figure 9 [4]. The thermal contact between two bodies is modelled with conforming and nonconforming meshes using linear (HEX8) and quadratic (HEX20) elements. Obviously the contact definition of the higher order elements is wrong, at least unsatisfactory. This is even more surprising, as the HEX20-element of this code shows very good performance when used in structural contact problems as shown in Figure 4. So any contact used in commercial codes needs to be checked at some benchmark problems before using it in engineering or scientific application.

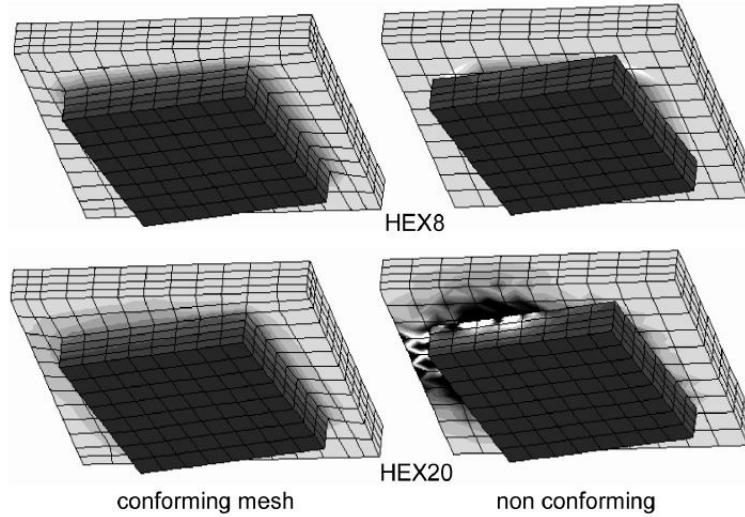


Fig. 9 Failing contact quality using HEX20 [11].

Scatter And Robustness

The increasing computing power improved the simulation of parts from single jobs of few variants to the study of the response of large sets of models. This may be necessary when we need to vary some or many of the parts parameters. A typical application is the modelling of the scatter of manufactured components in large series production. The term of robustness entered the world of virtual development opening new sights, new possibilities and new problems to the simulation teams. At the onset of robustness studies due to the large number of parameter combinations it was considered to be available only for few elaborated problems. The introduction of statistically based approaches like the Latin Hypercube and the Response Surface allowed accelerations of the studies. So today we may handle many problems with scattering parameters in acceptable and affordable time. Nevertheless robustness studies, especially in the nonlinear regime need essentially more time and computing power. This is due to the fact that many variants have to be analysed over a certain time of the contact process.

- **Scatter In Contact Studies**

Table 1 offered a selection of parameters and solution strategies used in contact modelling. The selected values of the parameters and their values influence the result of the contact analysis. In consequence robustness or sensitivity studies seem inevitable. Table 3 lists some of the most dangerous and scattering components of the physical contact and its numerical modelling. Once more neither completeness nor uniqueness is assumed. Other analysts with other experiences may add or remove entries to or from Table 3. In any case care should be taken that the range of possible states is covered by the range of numerical parameters.

Table 3 Most sensitive and influencing parameters in contact modelling.

Physical data	Numerical data
– friction	– gap, catch range
– surface quality	– mesh quality
– surface definition	– element definition
– loads	– initial contact
– supports	– penetration
– flexibility	– local stiffness – friction model

- **Parameter Sensitivity**

Figures 6 to 9 indicate some of the possible uncertainties encountered using specific contact models. The range of input definitions is hardly understood by the majority of the engineers working with the different numerical codes. Figure 10 gives an idea how small input changes result in large differences in the results predicted by the simulation. As a consequence automated or semi-automated systems should provide the possibility to do a sensitivity study to understand whether physical or numerical effects influence the results of the nonlinear analysis.

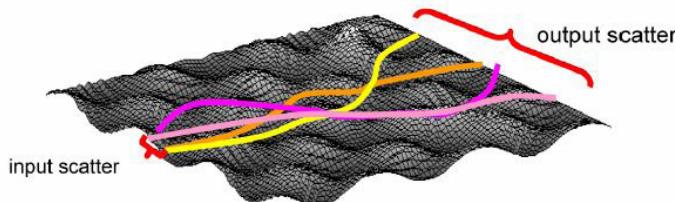


Fig. 10 Scatter and robustness.

Some typical steps may include:

- Check which input parameter changes cause significant changes in results.
- Cover possible parameter range.
- Find critical parameter combinations.
- Check reliability of parameters used.
- Compare results with experimental data, if available and reliable.

The outcome of these studies could result in an increased critical interpretation of the numerical results. This is always a good idea in nonlinear studies, especially in large and complex contact problems like crash and metal forming.

Conclusions

As a resume, some statements may help to improve the awareness of problems, obstacles and traps in contact simulation.

- Contact studies are possible with commercial and private simulation codes and without deep understanding of the underlying principles. Regarding the questions mentioned in this article, we doubt that this is always positive. Sometimes there should be some fences preventing inexperienced users to enter fields where their simulation is in great danger of being at least not very meaningful. The definition of these fences requires large experience.
- As many parameters for contact modelling are available in commercial simulation systems, many aspects have to be taken into account, many of the sometimes contradicting possibilities checked, interactions looked at, inconsistencies excluded.
- Often the meaning of parameters is not properly defined. Therefore a reproducible strategy of benchmarking and quality assurance needs to be established, improved and updated
- The tendency of using the default values provided by the code includes the danger to run totally inappropriate studies. All default values have to be checked, questionable entries removed, large warning signs posted to avoid following nonsense roads.
- The large number of physical and numerical parameters in contact studies makes a consequent system of robustness and sensitivity studies inevitable. It should be included in the problem solving rules and never be neglected due to lack of time or money.
- The interpretation of nonlinear simulations in most cases is a difficult task. So experience, comparison with experiments, preceding studies, comparable results have to be included in the evaluation of simulation results. This holds not only but even more thoroughly when dealing with contact problems.
- Qualified users should not and cannot avoid doing contact studies. They should always be aware of the many possibilities of failing by ignoring or misinterpreting physical or numerical aspects.

References

- [1] R. Steinbuch. Remarks on the engineering treatment of nonlinear simulation problems. Mathtool03, St. Petersburg, Russia, June 2003.
- [2] G. Kammler, G. Mauch, and R. Steinbuch. Automatische Neuvernetzung stark deformierter Strukturen. Berichtsband des MARC-Benutzertreffens 1992, München, 1992 [in German].
- [3] J.F. Molinari. Multiscale modelling of nanotribology: Challenges and opportunities. Presented at ICCC09, Lecce, 2009.
- [4] G. Zavarise and L. De Lorenzis. A strategy for contact problems with large initial penetrations. Presented at ICCC09, Lecce, 2009.
- [5] http://resources.altair.com/hyperworks/pdfs/product_brochures/HW13_HyperForm.pdf
- [6] Private communication with Michael Lautsch, Lautsch Finite Elemente, Esslingen, Germany.

19.15 Nonlinear Analysis Tutorials And Videos

In your HyperWorks Help Documentation you'll find the following tutorials:

NLSTAT:

OS-1360: NLSTAT Analysis of Gasket Materials in Contact

OS-1365: NLSTAT Analysis of Solid Blocks in Contact

Academic Training Center:

Exercise 3.1 Model Setup - Beam (NLSTAT)

Exercise 3.2 Contact Analysis - Bolted Flange (NLSTAT)

Exercise 3.3 Thermal Loading/Unloading - Beam (NLSTAT)

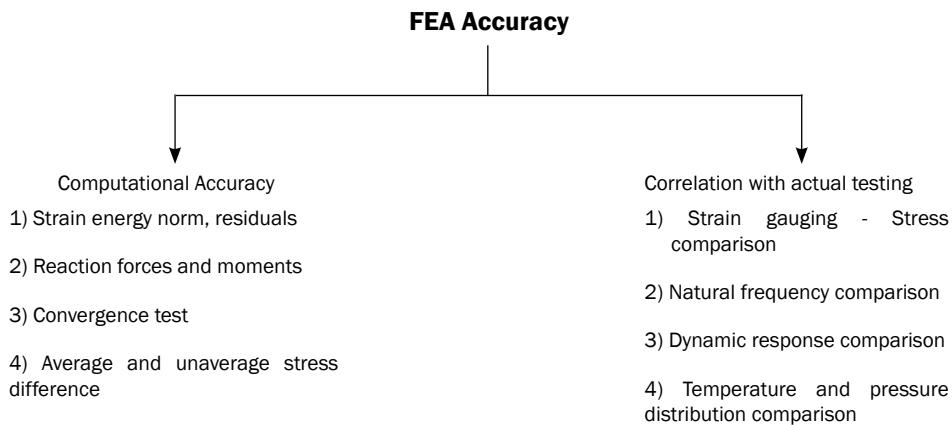
20 Post-Processing

This chapter includes material from the book “**Practical Finite Element Analysis**”. It also has been reviewed and has additional material added by Kristian Holm, Hossein Shakourzadeh and Matthias Goelke.

Please note that this chapter is mostly intended for post-processing of linear analysis results. Not all the comments made are relevant for crash or nonlinear analysis.

20.1 How To Validate And Check Accuracy Of The Result

Finite Element Analysis is an approximate technique. The level of accuracy of the displayed results could be 25%, 60%, or 90% with respect to the experimental data. The following checks helps in reducing the error margin



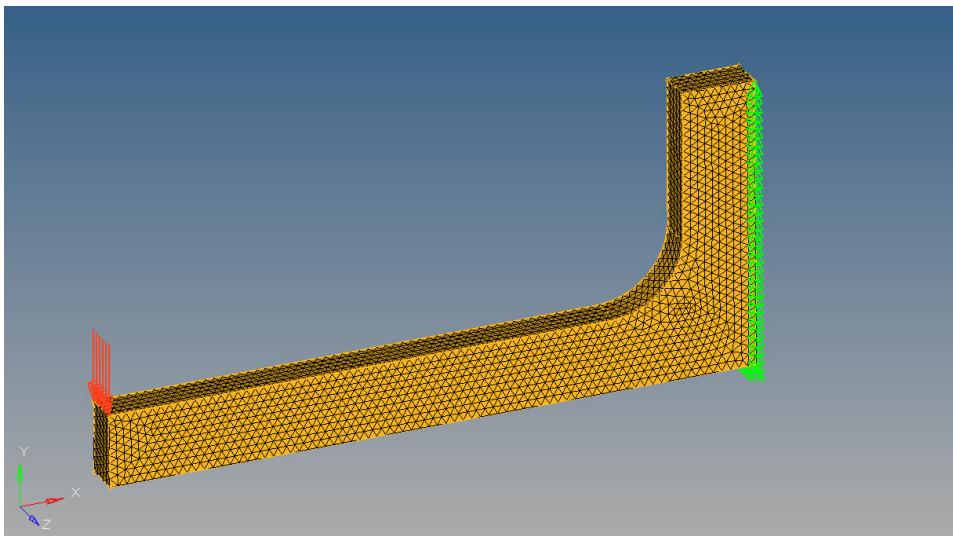
- Visual check – Discontinuous or an abrupt change in the stress pattern across the elements in critical areas indicate a need for local mesh refinement in the region.
- 10 to 15% difference in FEA and experimental results is considered a good correlation
- Probable reasons for more than 15% deviation – wrong boundary conditions, material properties, presence of residual stresses, localized effects like welding, bolt torque, experimental errors etc.

Computational accuracy does not guarantee the correctness of the Finite Element Analysis (i.e. the component may still behave in a different way in reality, than what's being predicted by the software). Correlation with test results or approval of the results by an experienced CAE / Testing department engineer (who has worked on a similar product / component over the years) is necessary. One has to distinguish between the FE error due to the quality of mesh and the deviation of the mathematical model to the physical problem related to modeling assumptions

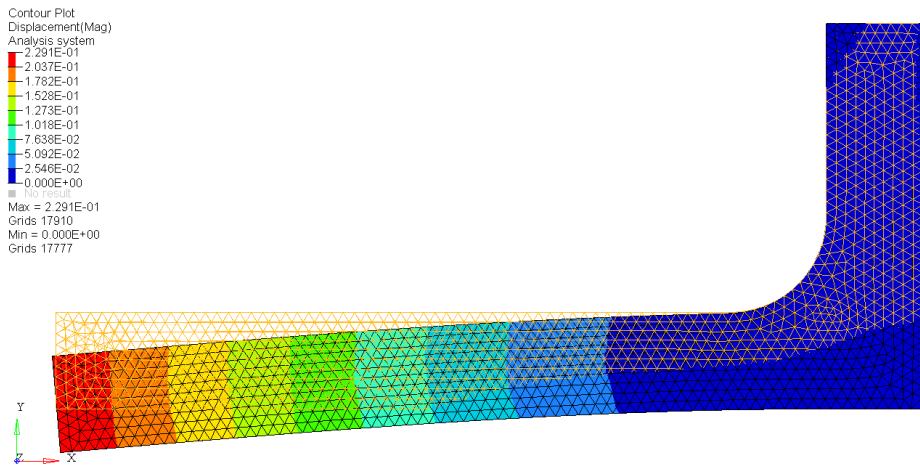
20.2 How To View And Interpret Results

Important Rule Of Thumb

Always view the displacement and animation for deformation first, and then any other output. Before viewing the result please close your eyes and try to visualize how the object would deform for the given loading conditions. The deformation given by the software should match with this. Excessive displacement or illogical movement of the components indicate something is wrong.

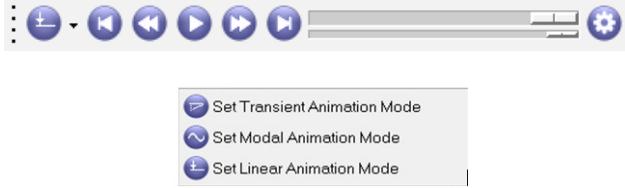


The displacements shown in the plot below is exaggerated to be able to see the results properly. The true (scalefactor = 1.0) displacement might not be visible at all due to a very small magnitude. Therefore, most of the post processors provide the ability to display scaled results (without actually changing the magnitude of the results).



Above: Displacement contour plot; displacements scaled by factor 100.

An extremely useful visualization technique is to animate results. This option is also (and especially) useful while interpreting results from a static analysis. The animated motion of the model provides insight into the overall structural response of the system due to the applied loads (and constraints).



Check Reaction Forces, Moments, Residuals And Strain Energy Norms

Comparing the summation of applied forces or moments and reaction forces or moments, external and internal work done, and residuals helps in estimating the numerical accuracy of the results. The values should be within the specified limits. This can be activated by selecting Check > Loads Summary (under the OptiStruct UserProfile).

The following summary table about the applied loads can be output:

LOADS SUMMARY												
15:31:13 - 02/07/2012												
ResultSys,0												
Number of forces,6												
Number of moments,0												
Number of pressures,0												
*SUM												
Node ID	X	Y	Z	Fx	Fy	Fz	Mx	My	Mz	F	M	
SUM	0.000	0.000	0.000	0.000e+000	-6.000e+002	0.000e+000	7.500e+003	0.000e+000	0.000e+000	6.000e+002	7.500e+003	
*LOAD COLLECTORS												
Forces,2												
*APPLIED FORCES												
Node ID	X	Y	Z	Fx	Fy	Fz	F					
17910	0.000	50.000	0.000	0.000e+000	-1.000e+002	0.000e+000	1.000e+002					
18992	0.000	50.000	25.000	0.000e+000	-1.000e+002	0.000e+000	1.000e+002					
20349	0.000	50.000	20.000	0.000e+000	-1.000e+002	0.000e+000	1.000e+002					
20350	0.000	50.000	15.000	0.000e+000	-1.000e+002	0.000e+000	1.000e+002					
20351	0.000	50.000	10.000	0.000e+000	-1.000e+002	0.000e+000	1.000e+002					
20352	0.000	50.000	5.000	0.000e+000	-1.000e+002	0.000e+000	1.000e+002					

Results data such as SPCFORCE etc. must be requested through the GLOBAL_OUTPUT_REQUEST (Setup > Create > Control Cards).

ACMODL	DEBUG	ECHO	
ANALYSIS	DENSITY	EIGVNAME	
BZGG	DENSRES	ELEMQUAL	
BULK_UNSUPPORTED_CARDS	DESHIS	FORMAT	
CTRL_UNSUPPORTED_CARDS	DESVARG	GAPRNM	
CHECK	DMIGNAME	GLOBAL_CASE_CONTROL	
CONTPRM	DTI_UNITS	GLOBAL_OUTPUT_REQUEST	

Card Image	X																													
<table border="1"> <tbody> <tr> <td>SPC FORCE</td> <td>FORMAT(1) H3D</td> <td>SPCF_FORM(1)</td> <td>SPCF_TYPE(1)</td> <td>OPTION(1) ALL</td> <td></td> </tr> <tr> <td>STRESS</td> <td>SORTING(1)</td> <td>FORMAT(1) H3D</td> <td>FORM(1)</td> <td>TYPE(1) ALL</td> <td>LOCATION(1) CORNER</td> </tr> <tr> <td></td> <td></td> <td></td> <td></td> <td>RANDOM(1)</td> <td>OPTION(1) YES</td> </tr> </tbody> </table>	SPC FORCE	FORMAT(1) H3D	SPCF_FORM(1)	SPCF_TYPE(1)	OPTION(1) ALL		STRESS	SORTING(1)	FORMAT(1) H3D	FORM(1)	TYPE(1) ALL	LOCATION(1) CORNER					RANDOM(1)	OPTION(1) YES	delete											
SPC FORCE	FORMAT(1) H3D	SPCF_FORM(1)	SPCF_TYPE(1)	OPTION(1) ALL																										
STRESS	SORTING(1)	FORMAT(1) H3D	FORM(1)	TYPE(1) ALL	LOCATION(1) CORNER																									
				RANDOM(1)	OPTION(1) YES																									
	disable																													
	enable																													
	next																													
	return																													
Card Image	X																													
<table border="1"> <tbody> <tr> <td>SPC FORCE</td> <td>FORMAT(1) H3D</td> <td>SPCF_FORM(1)</td> <td>SPCF_TYPE(1)</td> <td>OPTION(1) ALL</td> <td></td> </tr> <tr> <td>STRESS</td> <td>SORTING(1)</td> <td>FORMAT(1) H3D</td> <td>FORM(1)</td> <td>TYPE(1) ALL</td> <td>LOCATION(1) CORNER</td> </tr> <tr> <td></td> <td></td> <td></td> <td></td> <td>RANDOM(1)</td> <td>OPTION(1) YES</td> </tr> <tr> <td><input type="checkbox"/> ACCELERATION</td> <td><input type="checkbox"/> DISPLACEMENT</td> <td><input checked="" type="checkbox"/> SPCF</td> <td>SPCFORCES_NUM = <input type="text" value="1"/></td> <td></td> <td>reject default</td> </tr> <tr> <td><input type="checkbox"/> STRAIN</td> <td><input checked="" type="checkbox"/> STRESS</td> <td>STRESS_NUM = <input type="text" value="1"/></td> <td></td> <td>abort return</td> </tr> </tbody> </table>	SPC FORCE	FORMAT(1) H3D	SPCF_FORM(1)	SPCF_TYPE(1)	OPTION(1) ALL		STRESS	SORTING(1)	FORMAT(1) H3D	FORM(1)	TYPE(1) ALL	LOCATION(1) CORNER					RANDOM(1)	OPTION(1) YES	<input type="checkbox"/> ACCELERATION	<input type="checkbox"/> DISPLACEMENT	<input checked="" type="checkbox"/> SPCF	SPCFORCES_NUM = <input type="text" value="1"/>		reject default	<input type="checkbox"/> STRAIN	<input checked="" type="checkbox"/> STRESS	STRESS_NUM = <input type="text" value="1"/>		abort return	reject default
SPC FORCE	FORMAT(1) H3D	SPCF_FORM(1)	SPCF_TYPE(1)	OPTION(1) ALL																										
STRESS	SORTING(1)	FORMAT(1) H3D	FORM(1)	TYPE(1) ALL	LOCATION(1) CORNER																									
				RANDOM(1)	OPTION(1) YES																									
<input type="checkbox"/> ACCELERATION	<input type="checkbox"/> DISPLACEMENT	<input checked="" type="checkbox"/> SPCF	SPCFORCES_NUM = <input type="text" value="1"/>		reject default																									
<input type="checkbox"/> STRAIN	<input checked="" type="checkbox"/> STRESS	STRESS_NUM = <input type="text" value="1"/>		abort return																										
	abort return																													

Then, in the corresponding *.out file the following summary is provided:

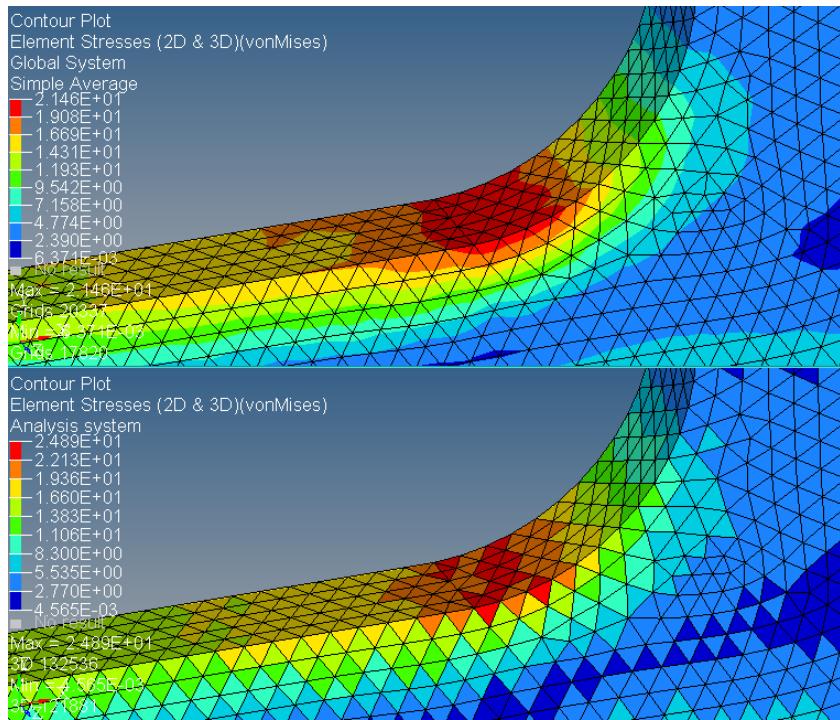
```
Edit Out File D:/home/goelke/test/bracket_12/bracket.out
File Edit
***** ANALYSIS RESULTS : *****

ITERATION 0
(Scratch disk space usage for starting iteration = 14 MB)
Residual energy ratio for static load case 1 = -4.150654E-10
Label x-force y-force z-force x-moment y-moment z-moment
Sum-App. 0.000E+00 -6.000E+02 0.000E+00 7.500E+03 0.000E+00 -2.132E-11
Sum-SPCF 2.389E-10 6.000E+02 3.737E-10 -7.500E+03 3.638E-08 -1.995E-05

Notes: 1. All applied and SPC forces are transferred to
       the origin of the basic coordinate system,
       so that the applied and SPC loads can match.
2. If spring elements and/or MPCs exist in the model,
   total applied loads may not match total SPC loads.
(Running in-core solution)
```

Stress Plot

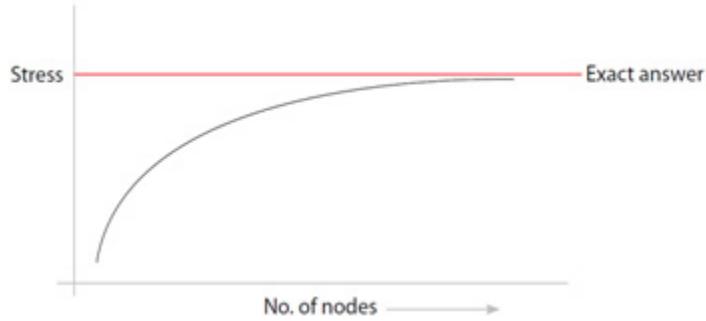
The location and contour in the vicinity of the maximum stress should be observed carefully. Discontinuities, or abrupt changes, in the stress pattern across the elements in critical area indicates the need for local mesh refinement. Commercial software offer various options for stress, like nodal, elemental, corner, centroidal, gauss point, average and unaverage, etc. Unaverage, corner, or nodal stress values are usually higher than the average, centroidal or elemental stress values.



4) Which stress one should refer to? If you were to ask this question to FEA experts from different companies, nations, or commercial software representatives, and you will be surprised to hear different answers; everyone confident about the practice that is followed by his or her company or software. The best way to understand the output options of a specific commercial software and to decide which stress one should be refer to is to solve a simple problem with a known analytical answer (like a plate with a hole) and compare the analytical answers with the various options.

Interestingly, if the same results are viewed in different post processors, it would show different result values. This is due to the software's default settings (some software default settings is to average the stress while others is to not average, some prefer elemental while other's nodal, etc.).

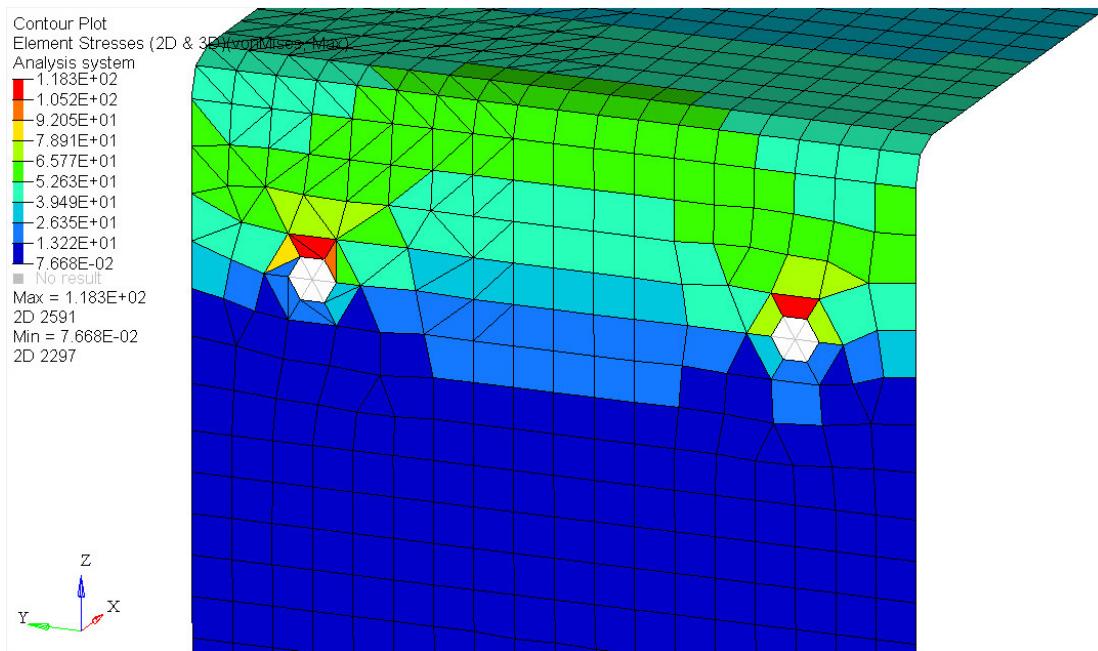
Convergence Test



In general, increasing the number of nodes improves the accuracy of the results. But at the same time, it increases the solution time and cost. Usual practice is to increase the number of elements and nodes in the areas of high stress (rather than reducing the global element size and remeshing the entire model) and continue until the difference between the two consecutive results is less than 5 to 10%. In the case of application of point forces on a FE model or the application of a boundary condition on a node, a high stress value can be observed. Refining the mesh around this point is not a solution as the theoretical stress value is infinite. See also the video of Prof. Chessa about "Convergence of Finite Elements" (<http://www.youtube.com/watch?v=YRJ0c8uO8iU>).

Mesher For Symmetric Structures Should Be Symmetric Otherwise The Analysis Would Show Unsymmetric Results (even for symmetric loads and restraints)

In the above figure stress is higher at one of the hole though the loading, restraints, and geometry are symmetric. This is due to meshing that was carried out using the auto meshing option. It created an unsymmetric mesh even though an equal number of elements were specified on both of the holes.

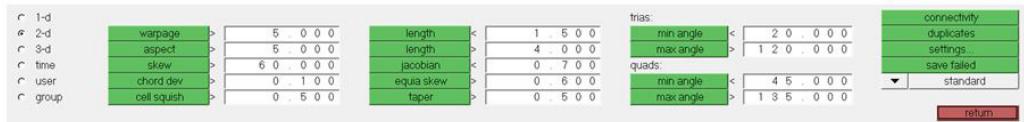


Importance Of Duplicate Element Check

Always perform a duplicate elements check before running the analysis: Duplicate elements are very dangerous and might go undetected (if scattered and not on the outer edge or boundary of the structure) in the free-edge check. Duplicate elements add extra thickness at respective locations and result in too less stress and displacement (without any warning and error during the analysis).



To check for duplicate elements activate the “Check” panel .



By the way, we only talk about duplicate elements, if these elements share the same nodes! Elements which lie on top of each other (but do have different nodes) are not duplicated. Please think this definition over!

Selection Of Appropriate Type Of Stress:

- VonMises stresses should be reported for ductile materials and maximum principal stresses for brittle material (casting) components.
- For nonlinear analysis, we should pay attention to true and engineering stress.
- True stress: is defined as the ratio of the applied load to the instantaneous cross-sectional area
- With engineering stress, the cross-section remains constant.
- Also see <http://dolbow.cee.duke.edu/TENSILE/tutorial/node3.html>)

Additional Questions To Ask

The following questions are also valuable when validating a linear analysis after computation:

- Is the maximum stress less than yield stress?
- Are the displacements are small with respect to the characteristic structural size?
- Is there any rotation larger than 10°?
- Is there any overlaps after deformation (contact)?

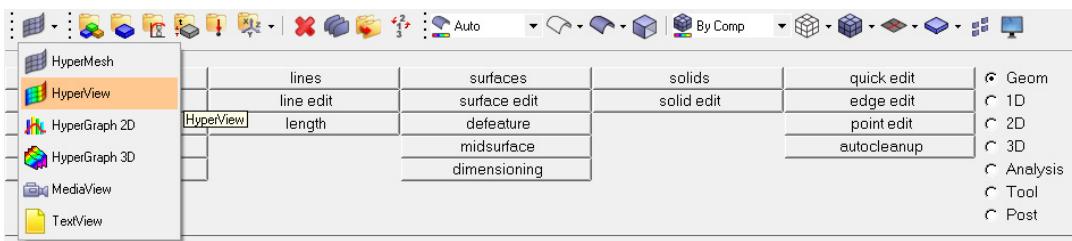
20.3 Post-Processing In HyperView

The open architecture of HyperView allows for loading and viewing result files obtained from several sources. Based on the solver type of the files and the results you would like to visualize and analyze, there are different ways to load the input deck and their corresponding results into HyperView. This chapter guides you through the various ways you can load your files and the various tools available for viewing the model according to your interest.

To begin post processing, in the HyperWorks Desktop Version split the screen:



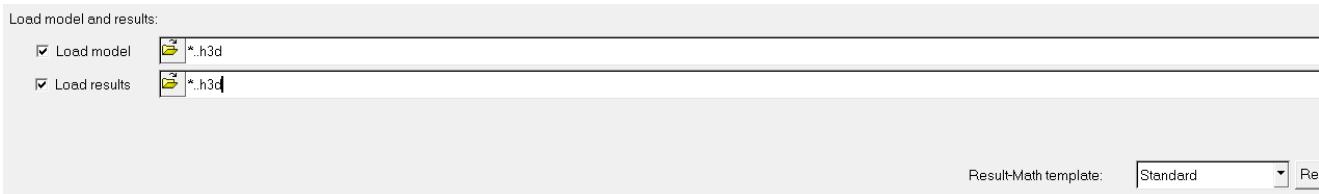
and then activate the client HyperView.



To access the Load Model panel:

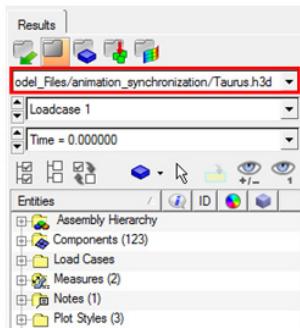
- Click the Open Model button  from the Standard toolbar.
- or
- From the menu bar, select File > Open > Model

The **Load Model** panel allows you to load the result files along with the model files. If the result file already contains the model definition, it is not a requirement that you load the model file along with the results. However, when only result files are loaded, the component definitions such as name and color are not preserved. The solver definition for component names along with the default color settings is loaded. You can also choose to load only a model or result file.



There are a couple of options in this panel: Overlay, Result math template, and Reader Options. The Result math template option allows you to select a template to be loaded into the Derived Result panel. The options are Standard, Advanced, NVH, Composite or None. The Reader Options button opens a window where different options can be specified for the different results readers.

Activating the **Overlay** check box in the panel allows you to load multiple models and their results into a single window. You can then set the active model in the window from the Results Browser. This is done by selecting the model from the model list in the Results Browser:



Supported Solver Result File Formats

The following solver result file formats are supported by HyperView:

HyperMesh results (RES) file	Hyper3D (H3D) file
Altair FLX file	Altair MRF file
ABAQUS ODB file	NASTRAN OP2 file
LS-DYNA D3PLOT and PTF files	ADAMS GRA and RES files
ANSYS RST and RTH files	DLM or LS-DYNA DYNAIN files
FEMZIP DSY file	DYNA DB file
MADYMO KIN3 (KN3) and FAI files	MARC T16 file
NIKE3D N3PLOT file	OptiStruct OP2 file
PAM-CRASH DSY file	RADIOSS A001 file
DADS BIN file	Moldflow UDM file
Universal UNV file	

In addition to the solver result file formats supported through direct readers, HyperView supports additional solver formats via result translators.

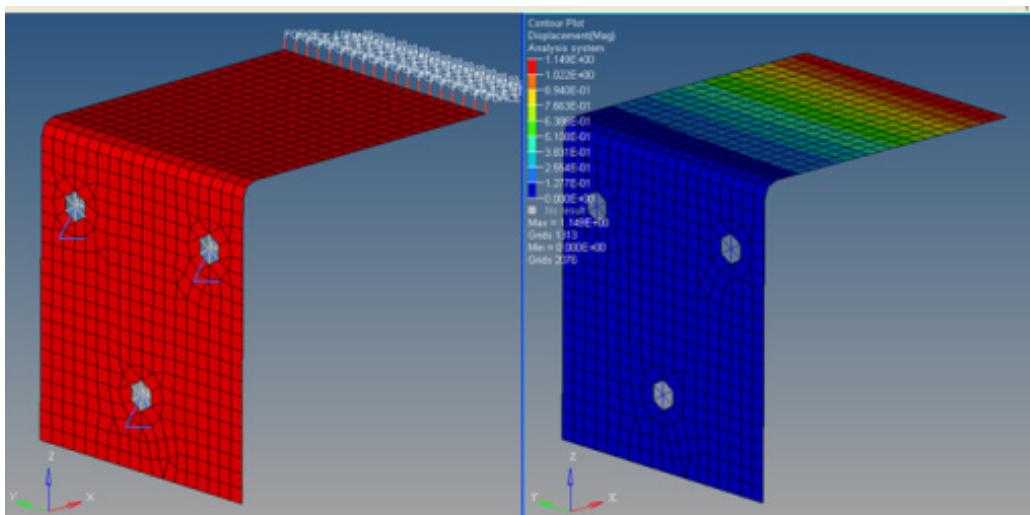
After reading in the results file (for example, *.h3d) have a look at the global displacements.



Activate the icon and specify the Result type: as Displacement.

Ask yourself:

- Are the displayed magnitudes reasonable?
 - Magnitudes in the order of e.g. 10e5 indicate that the model is pursuing a rigid body motion —> constraints are erroneous.
- Check the constraints in some detail
- Is the “response” of the model “correct”?



Tip: Animate your results—even though it was a static analysis, and check the global and local (where the constraints are placed) model behavior.



HyperWorks 11.0

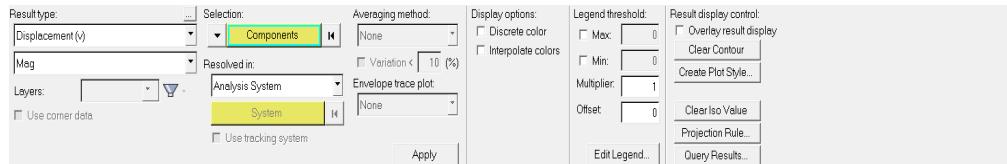
General Remark: FEM programs do not check whether the input data are meaningful. For instance, there may be typos in the Young's modulus, element thickness, magnitude of your loading, wrong constraints, mesh “mistakes” etc. As long as the FEM can solve the equations — you will get a result, regardless of whether it is correct or not.

Contour Plots

A contour plot generates color bands on the model, based on the values found in the results file. A contour plot can be created from tensor, scalar, vector, or complex results. There are two different ways to contour results in HyperView; the Contour panel and the Results Browser. There are advantages to using each tool. Below an in depth look at each tool is shown and the advantages are discussed.

The Contour Panel

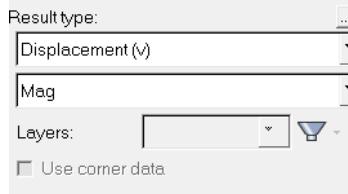
The “contour” panel allows you to contour a model and graphically visualize the results. In the contour panel you can view vector, tensor, or scalar type results. To access the contour panel either click the contour panel button  on the results toolbar, or select Results > Plot > Contour from the menu bar.



This panel, like most panels in HyperView, works from left to right. First the result type is defined, then the selection and resolved in system is selected, and finally the averaging method is defined. After these setting, the display and legend options are defined.

Result Type ...

From the drop down menus under the result type field, vector, tensor, or scalar result types are selected. First the result type (displacement, etc) is selected, then the component of the result type is selected (X, Y, Mag, etc). There is also an option, Entity with layers, that allows you to display a contour for a specified element layer when a layer definition is available for an element. The final option available under the result type heading is the Use corner data checkbox. This option is only active when corner data is available in the results file. When this option is selected, HyperView displays color bands by interpolating available corner results within each element. This allows for a discontinuity of the result distribution across element boundaries to be seen.



Selection ...

Next the selection on which the contour should be applied is selected. Before creating a contour plot, you may pick one or more entities from the model. You can do this by picking entities directly from the screen, using the quick window selection, or clicking the Elements, Components, or Assemblies input collector and using the extended entity selection menu. If no selection is made, the contour will be applied to displayed components or elements by default.

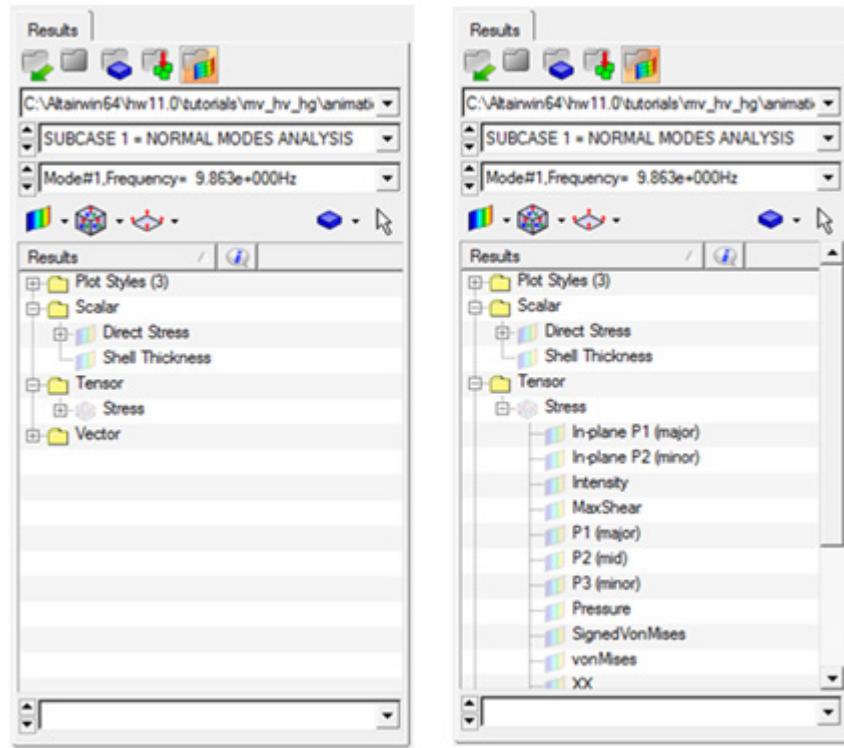
Resolved in ...

This drop-down menu allows you to select the result coordinate system to be used to contour the results. The available options are dependent on the current selection for the Averaging method. You can select the global, element, or analysis coordinate system as well as a user-defined system.

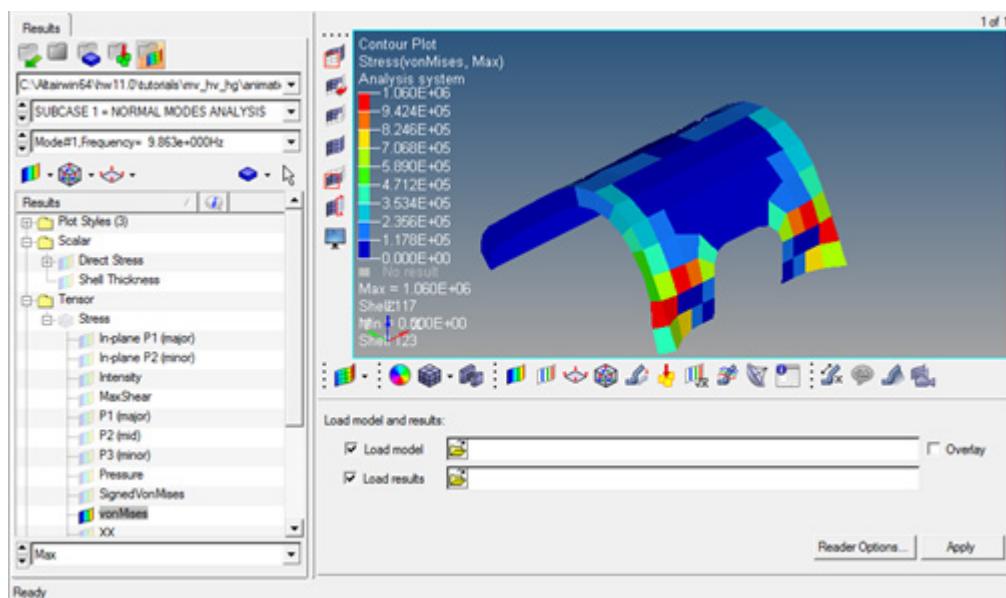
- Global System: Transforms to the global system.
- Elemental System: Transforms results to the element coordinate system. In HyperView, the element coordinate system is defined by element connectivity.
- Analysis System: Displays the vector and tensor results as they are output from the solver.
- User System: This option is available when the results file contains a user-defined coordinate system. The System input collector is enabled when User System is selected. Select a system by ID or pick from the screen.

The Results Browser

Within the Results Browser there is the Result View. The Result View shows a hierarchical view of available results for the current load case. The result types are grouped by their type, and are broken up into Scalar, Vector, and Tensor folders. You can expand the folder to see all of the details for each result type. For example, each of the Scalar, Tensor, and Vector folders are expanded to see the Result Type within the folders. By selecting one of those Result Types (for instance, Stress), the different Components are shown:



To apply a contour plot using the Results Browser, simply click on the icon to the left of the result to be used for the contour. For example, in the plot below, the vonMises component for Stress is contoured.

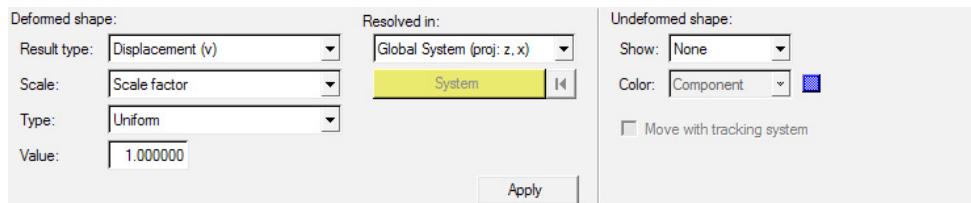


Viewing Deformed Shapes

The “deformed” panel allows you to specify parameters for deformation display. You can use this function to see the motion of your model after analysis. You can display the original structure and the deformed shape to see the total amount of movement, or view the deformed shape by itself. You can also create an animation sequence of the structure’s movement that shows the motion of the structure in a series of frames, based on what the analysis code has predicted the model will do.

To access the “deformed” panel:

- Click the Deformed panel button  on the Results toolbar.
- From the menu bar, select Results > Plot > Deformed



The first step in the Deformed panel is to define the Deformed shape. This includes defining the Result type and Scale factor to be applied. The Result types available in this panel depend on the result file that was imported. For example, all result files have Displacement (v) loaded as a Results type, but OptiStruct also has Rotation (v), Eigenvector (v), and Shape change (v) available.

Once the result type is determined, the scale factor needs to be assigned. The “Scale” can be set with respect to “model percent”, or “model units”. A scale factor multiplies the displacement to produce the deformed shape. This option is available for all animation modes. When the Model percent option is used, the deformed shape of the model is scaled, so that the maximum deformation of the model is displayed as a specified percentage of the current model size. The model size is the diagonal length of the axis-aligned bounding box which contains all model geometry. This is only available for modal and linear static animation modes. The final option, Model units, has the maximum value in the results displayed as the number of model units defined. This option is also only available for modal and linear static animation modes.

Next the Type of scaling to be done is selected. The two options are Uniform and Component. Uniform allows you to enter a single value into the Value field which is used to multiply each component (X, Y, and Z) by the same value. The Component options allows you to specify different scale factors for X, Y, and Z. To eliminate the movement in a direction, specify 0.0 in the component field for that direction.

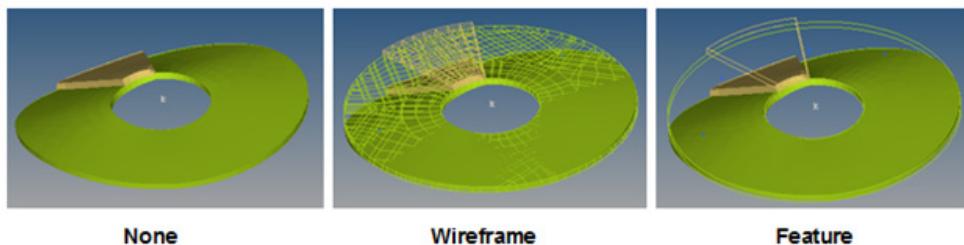
Once the Deformed shape has been specified, the Resolved in system is selected. The Resolved in drop down menu allows you to select the result system in which you want to display the results. The available options are:

- Global System: Transforms vectors into the global coordinate system.
- Analysis System: Displays the vector results as they are output from the solver.
- User System: Transforms vector results into a user defined system. This option is available when the results file contains a user defined coordinate system or a system has been created in HyperView.

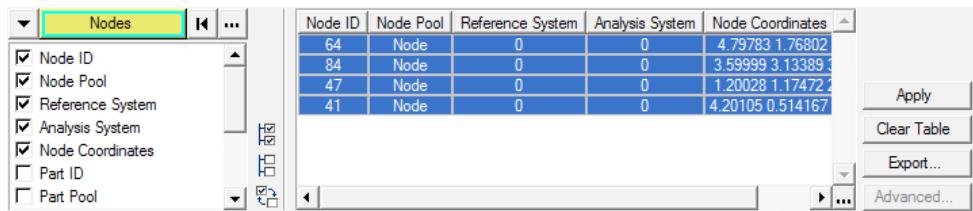
Once the “deformed shape” has been defined, the options for the Undeformed shape can be set. The Show option sets how the “undeformed shape” should be displayed:

- None: The deformed shape is not shown.
- Wireframe: The deformed shape is shown in wireframe mode.
- Edges: The edges of the deformed shape are shown.
- Features: The features of the deformed shape are displayed.

The color of the undeformed shape can also be defined. When Component or Mesh Lines is selected, the default colors are automatically applied to the undeformed shape. When User is selected, you can click the color button and select a different color for the undeformed shape.

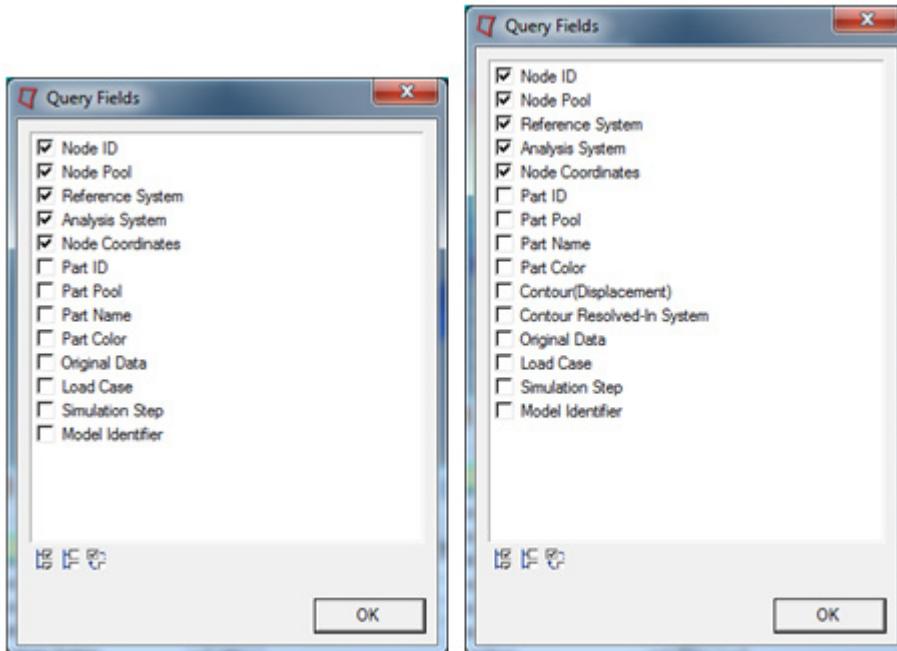


The Query panel allows you to view and export properties, as well as other information, for all nodes, elements, components, and systems contained in the active model. The Query panel can be accessed using the icon in the Results toolbar or by selecting Results > Query from the menu bar.



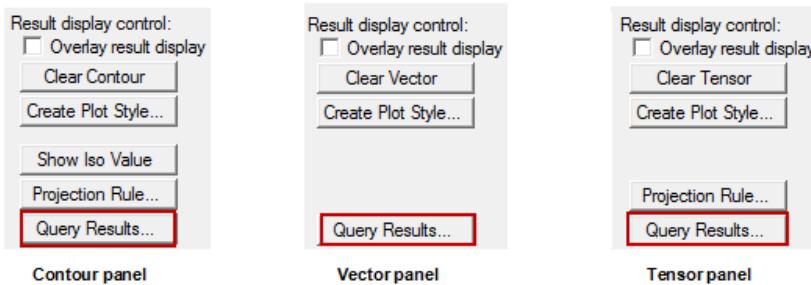
Nodes, elements, components, and systems can be selected to be queried. Depending on the entity type selected, different properties can be selected to be queried. Once the properties are selected, the entities can be selected by either graphically selecting them or using the extended entity selector. The table is populated as each entity is selected. To clear the table, the Clear Table button is selected. The "Export..." button is used to export the data in the table to a .csv file.

In order to query results, a contour, vector, or tensor plot needs to be applied to the model. Once a plot is created, the available properties for querying will appear in the property list. Also, by selecting the "..." after the entity selector, a full list of the properties available is opened. For example, the image to the left shows the properties available before a contour is applied, while the image to the right shows the properties available after the contour plot is created:



Notice how additional options are available for querying the contour results (Contour(Displacement) and Contour Resolved-in System). Another thing to note is that if the result type is nodal based, then the properties to query the results will only appear when Nodes are selected. The same is true with elemental results only being available when elements are selected. For a full listing of the available properties in the query panel, please refer to the Online Help.

Once the model has a contour, vector, or tensor plot applied, you can also access the Query panel directly from the Contour, Vector, or Tensor panel by clicking on the Query Results button located on the right side of each panel:

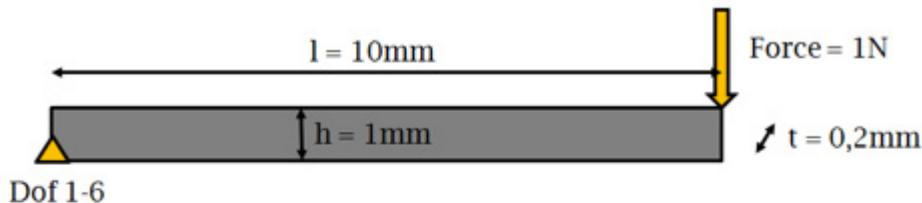


20.4 Stress Calculation And Output In Optistruct

This section is based on a webinar held by Kristian Holm. A link to the webinar is provided at the end of the section. In the examples below, the first paragraph focuses is on the standard/default stress output data, then the various averaging and extrapolation options available in OptiStruct and HyperView are discussed and explained.

Example: In-Plane Bending

The cantilever shown in the image below is meshed with shell elements. Note that Poisson's ratio NU, and Shear Modulus G are set to "0". Also the dimension of the cantilever is chosen such that the shear deformation effects can be neglected (Bernoulli beam).



Material: $E = 210.000 \text{ MPa}$
 $G, \text{NU} = 0$

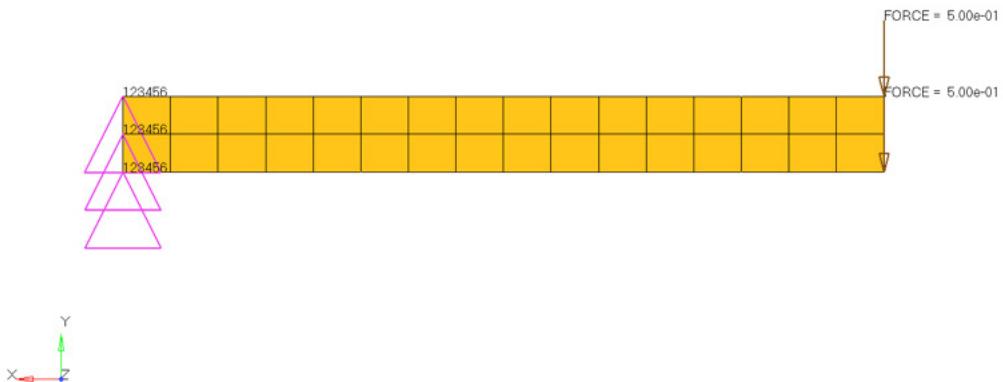
Theoretic results:

$$\sigma_{BM} = \frac{M_B}{I} \cdot \frac{h}{2} = 6 \cdot \frac{F \cdot l}{t \cdot h^2} = 300 \text{ MPa}$$

$$y_M = \frac{F \cdot l^3}{3 \cdot E \cdot I} = 4 \cdot \frac{F \cdot l^3}{E \cdot t \cdot h^3} = 0,095 \text{ mm}$$

with M_B bending moment ($F \cdot l$), I moment of area inertia ($t \cdot h^3/12$); σ_{BM} = bending stress at upper and lower left corner.

The file rods.fem contains the ready-to-run model. This model is meshed, has the appropriate constraints and loads applied, and the necessary cards are defined for analysis.



The displacement and stress results (based on the default output settings) are depicted in the following images:

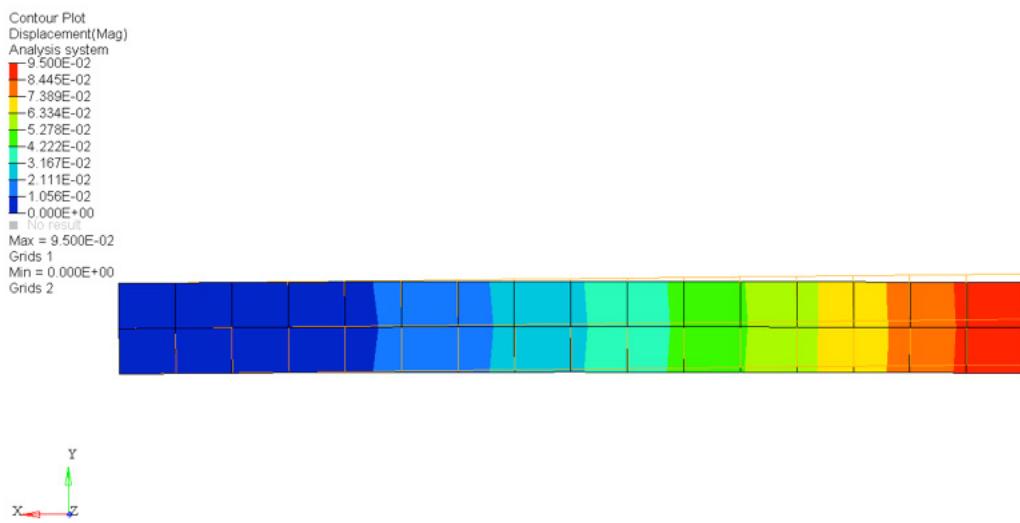
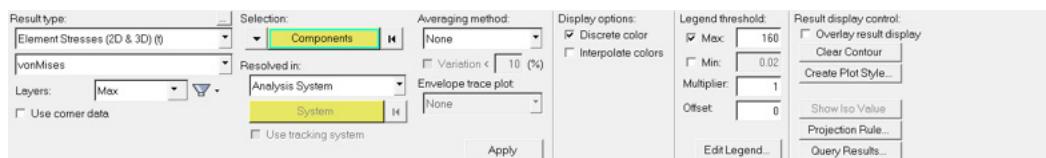


Figure: Displacement contour plot (displacements in mm)

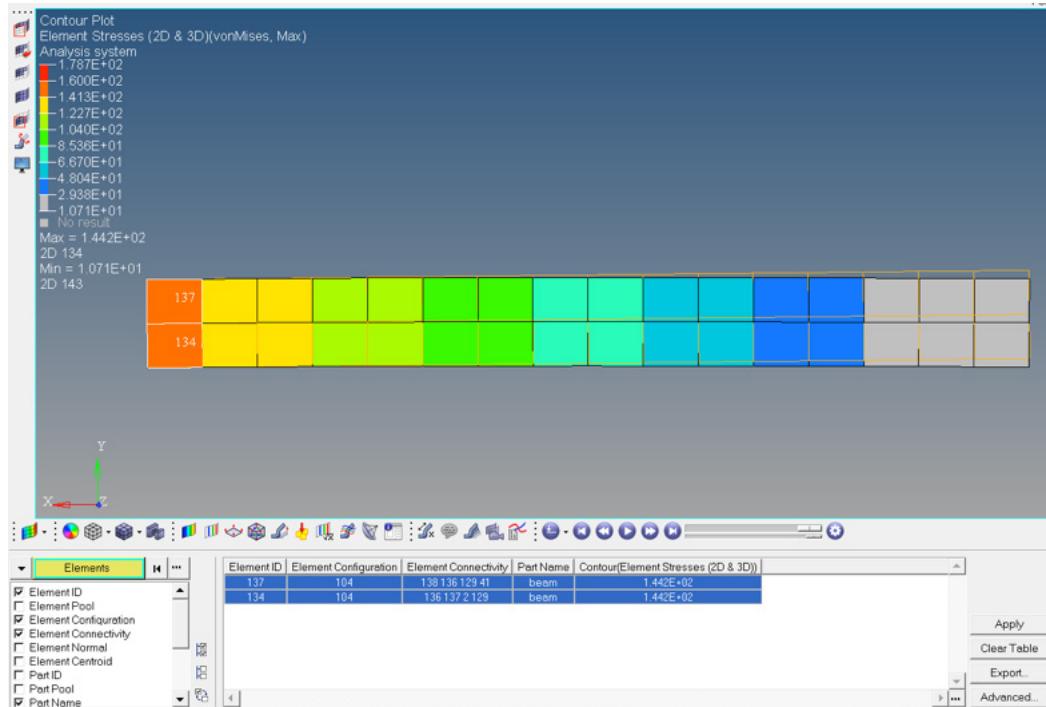
Apparently, the displacements match the analytical results pretty well.

However, what about stresses?

To better view/resolve the stresses, the legend threshold is adjusted to Max: 160 MPa:

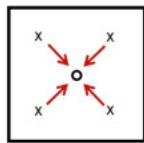


In the Query panel:



As listed in the Query panel, the stresses in the two elements (id 134 and 137) are 144.2 MPa, whereas the analytical value is 300 MPa.

Quite a difference! Why?



By default element stresses for shell and solid elements are output at the element center only.

x : Integration Points

o : Element center

Thus, the analytical stresses needs to be calculated with respect to the coordinates of the element centroid $h/4$ (and not at its upper or lower edge at $h/2$)

Theoretic result at element center

$$\sigma_{B_elementCentre} = \frac{M_B}{I} \cdot y = 12 \cdot \frac{F \cdot l}{t \cdot h^3} \cdot y$$

with $y = \frac{h}{4}$

$$\sigma_{B_elementCentre} = 3 \cdot \frac{F \cdot l}{t \cdot h^2} = 150 \text{ MPa}$$

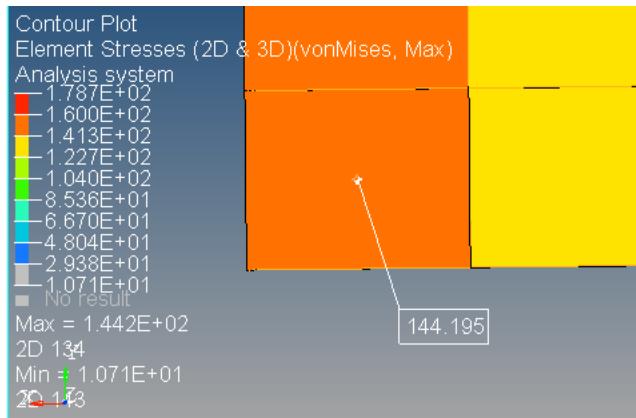
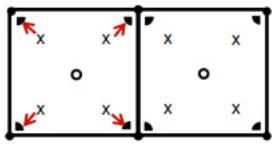


Figure: Stress at element centroid

The element centroid stress is 144 MPa, whereas the analytical value is 150 MPa indicating an error of about 4 %.

What about the stress at the elements “upper / lower edge”?

Hereto, the stresses at the integration points are extrapolated to the element nodes.



- x: Integration Points (IP)
- o: Centroid
- : Element nodes
- ▲: „Corner data“ = nodal results (participation of element to a node)

As depicted in the image, extrapolation of the integration point stresses takes places within each element. Thus, depending on the number of elements attached to a particular node, different stress values are associated with a single node, respectively.

This may cause numerical stress “jumps” from element to element. These “jumps” strongly depend on the kind of extrapolation chosen and the mesh size.

There are different extrapolation options available. The simplest one is bilinear extrapolation (always with respect to the stresses at the integration points).

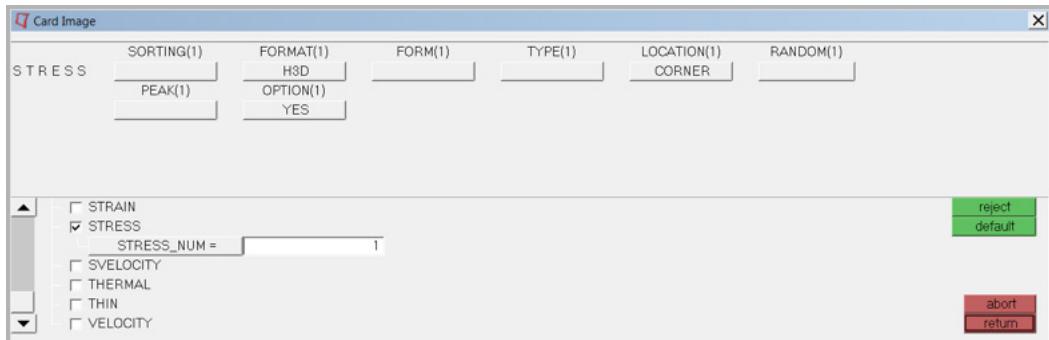
Note, the kind of extrapolation needs to be specified in OptiStruct before the analysis is started! BILINEAR extrapolation of stress data can be requested through either:

STRESS(H3D,CORNER) = YES

STRESS(H3D,BILIN) = YES

Analysis > control cards > GLOBAL_OUTPUT_REQUEST

GAPPRM	INCLUDE_CTRL	M2GG	delete
GLOBAL_CASE_CONTROL	INFILE	MECHCHECK	disable
GLOBAL_OUTPUT_REQUEST	K2GG	MODEL	enable
GRDSET	K2PP	Model Documentation	
GROUNDCHECK	K42GG	MSGLMT	
HISOUT	LABEL	OMIT	next
INCLUDE_BULK	LOADLIB	OSDIAG	prev
			return



Only if this request has been made inside the solver, then the Use corner data option is active in HyperView (Contour panel).

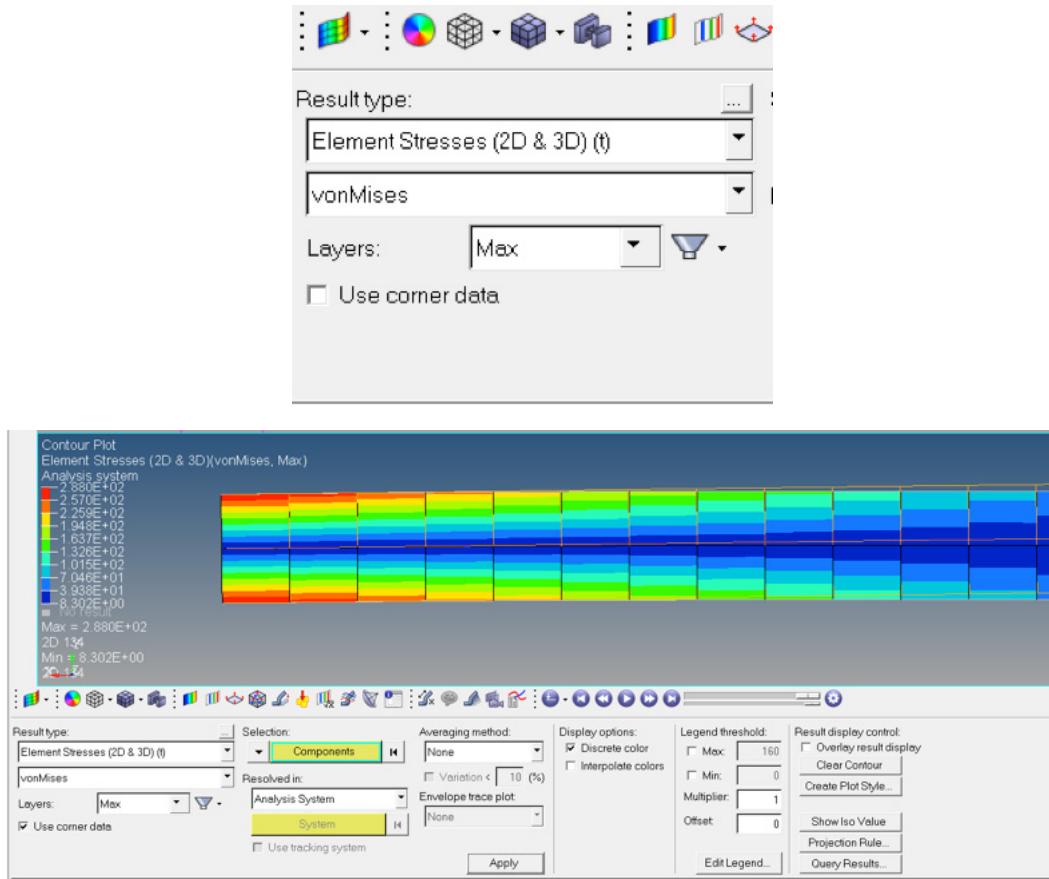
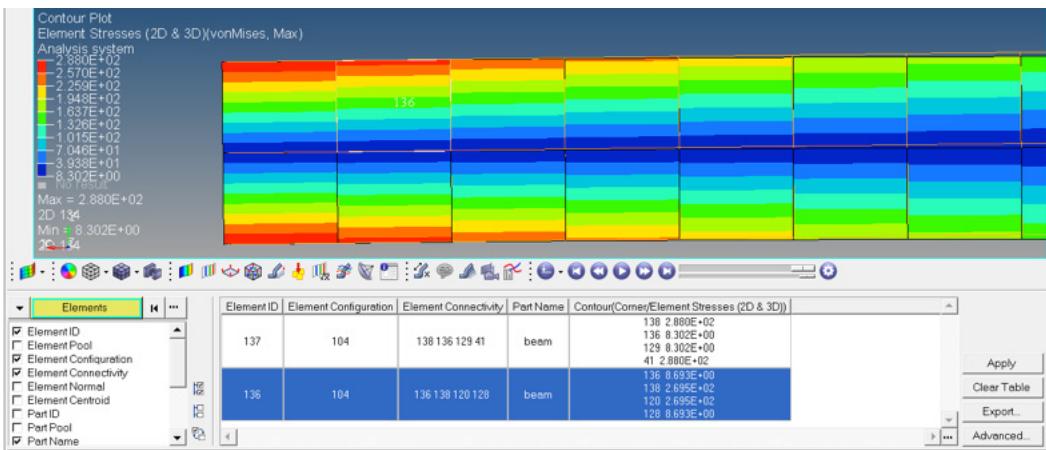


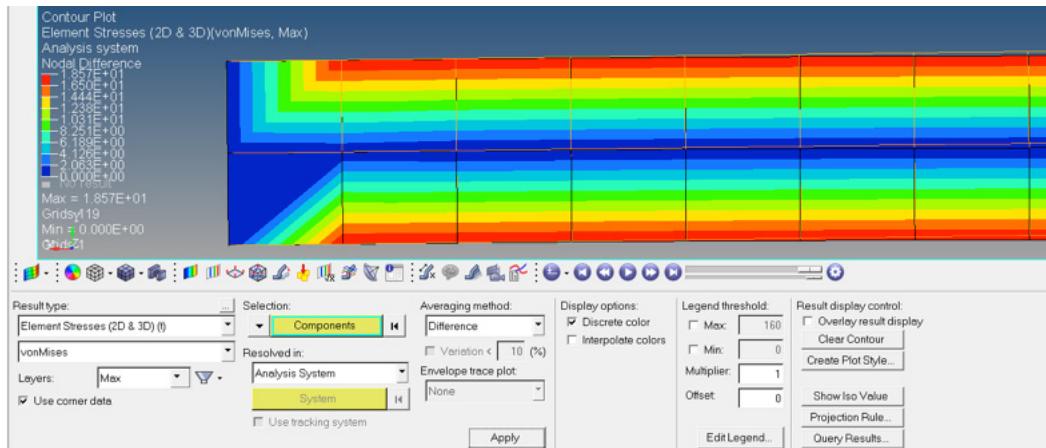
Figure: Element stress contour plot based on bilinear extrapolation. In the contour panel “Use corner stress” is activated



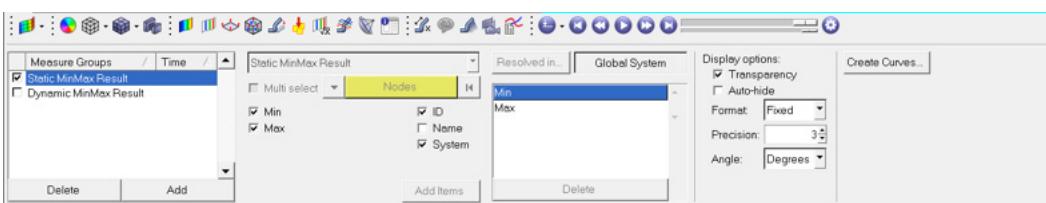
The query option now lists 4 different stress results per element. In element 137, the stress at node 138 and 41 is 288 MPa, at node 136 and 129 the stress is 8.3 MPa. In contrast, in element 136 the stress at node 138 is 269 MPa (i.e. two different stress results at the same node).

Quite obviously, the finer the mesh discretization in the longitudinal direction, the smaller the stress jumps from element to element will be.

To better visualize the stress difference across adjacent elements, inside HyperView the averaging method Difference may be used. This averaging method simply calculates the difference between maximum and minimum stress value at each node. This kind of contour plot thus can also be considered (or used) as a measure of element discretization quality.

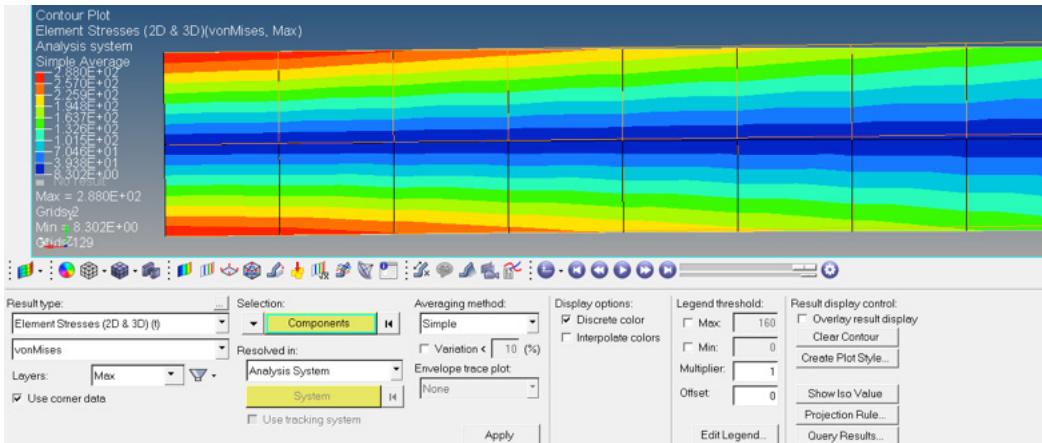


Making use of the max/min option (in the Measure panel) helps to identify the location of the greatest stress gradient between adjacent elements:

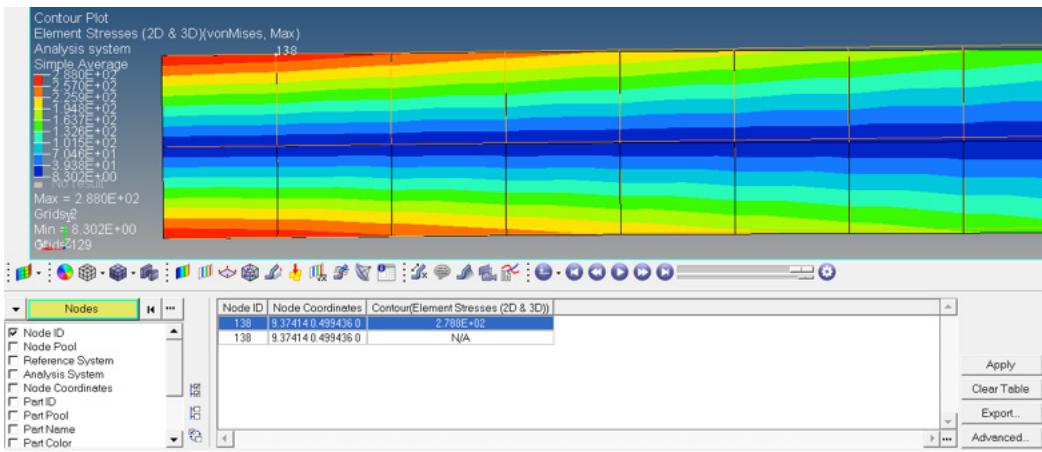


If the maximum stress gradient is located in a critical area of the structure local remeshing may be required.

Another averaging method is named **Simple**. This method averages the various stress results per node:



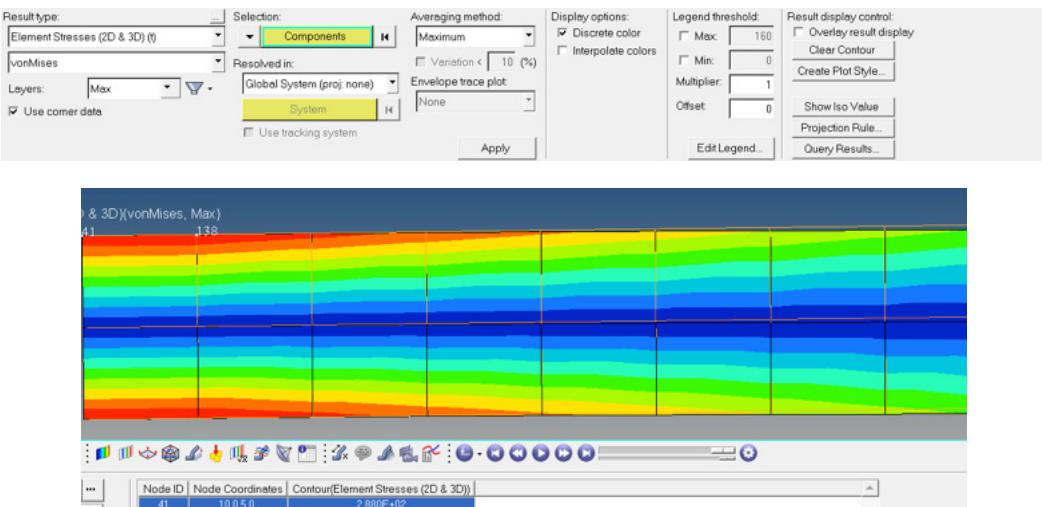
The query option for node 138 lists:



278.8 MPa (compare with the values shown before). However, this method “simply” averages the given von Mises stress (which is not entirely correct). More precise is to averaging the stress tensor first and then to form the von Mises at a common node.

This can be accomplished through the averaging method named Advanced. In this case since we are looking at a uniaxial stress state simple and advanced are the same. In general **Advanced** is recommended over **Simple**.

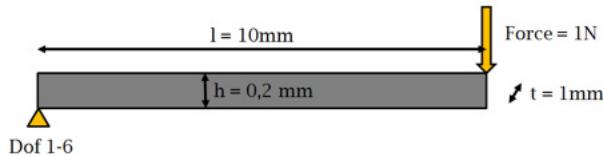
If the averaging method **Maximum** is used, then the maximum value per node is displayed, respectively.



This averaging method leads to a more conservative assessment of the structural behavior.

Summary: Using the output options BILINEAR during the solver run (analysis) leads to more “realistic” values.

Example: Out Of Plane Bending



Material: $E = 210.000 \text{ MPa}$
 $G, \nu = 0$

Theoretic results:

$$\sigma_{BM} = \frac{M_B}{I} \cdot \frac{h}{2} = 6 \cdot \frac{F \cdot l}{t \cdot h^2} = 1500 \text{ MPa}$$

$$y_M = \frac{F \cdot l^3}{3 \cdot E \cdot I} = 4 \cdot \frac{F \cdot l^3}{E \cdot t \cdot h^3} = 2,38 \text{ mm}$$

Note that in this model the width of the beam is 1 mm its height is 0.2 mm.

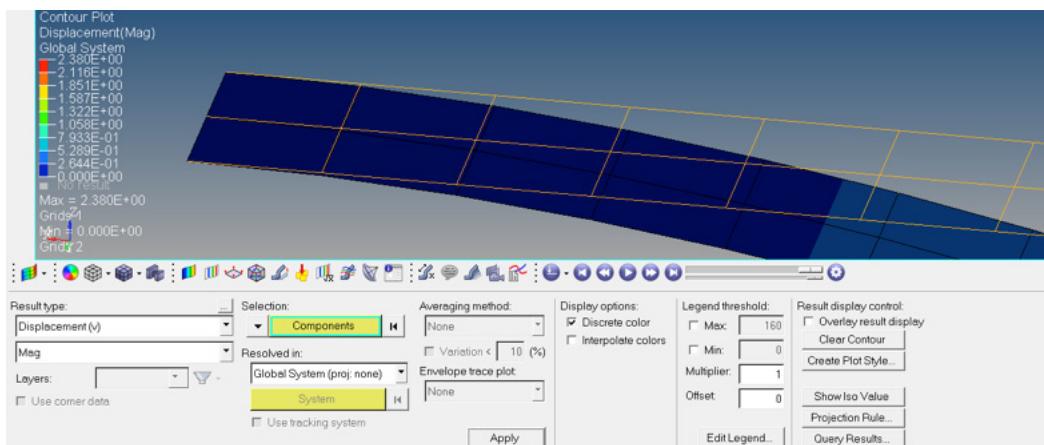


Figure: Displacement contour plot.

The displacements of 2.38 mm again match the analytical result.

What about the stresses?

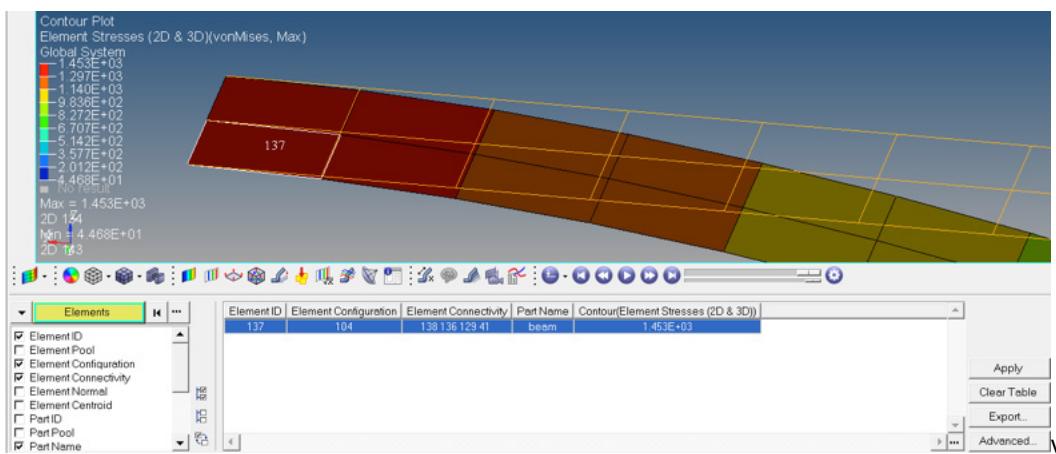
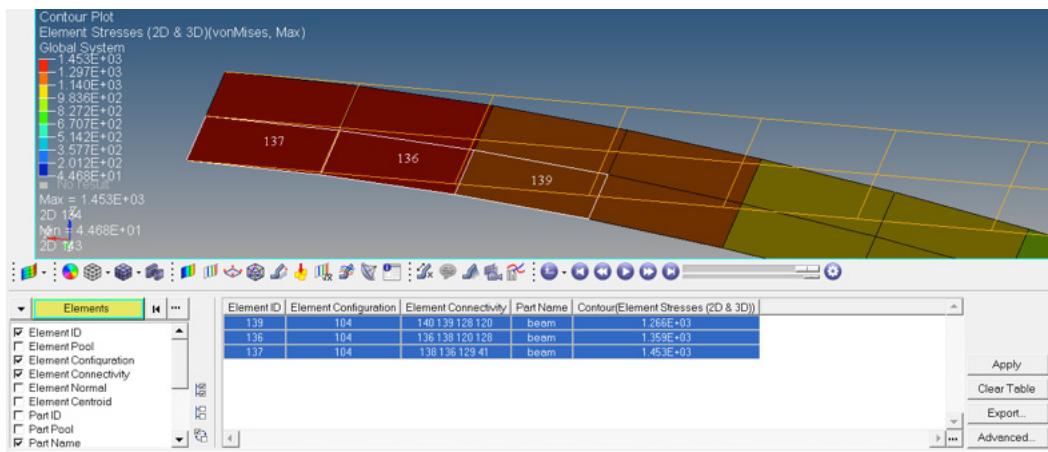


Figure: von Mises stress at element centroid

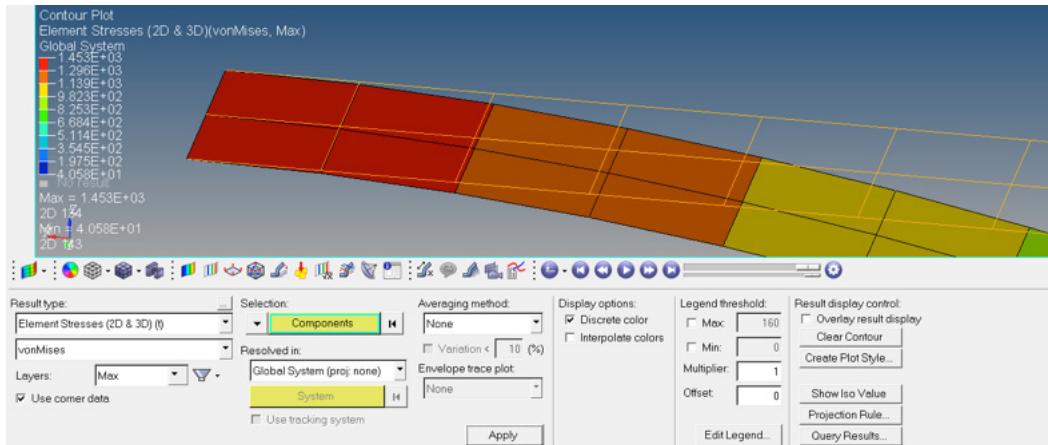
The stress at the element centroid is 1453 MPa.

Looking at the three adjacent elements, we see a jump by about 100 MPa from element 139 to element 136.

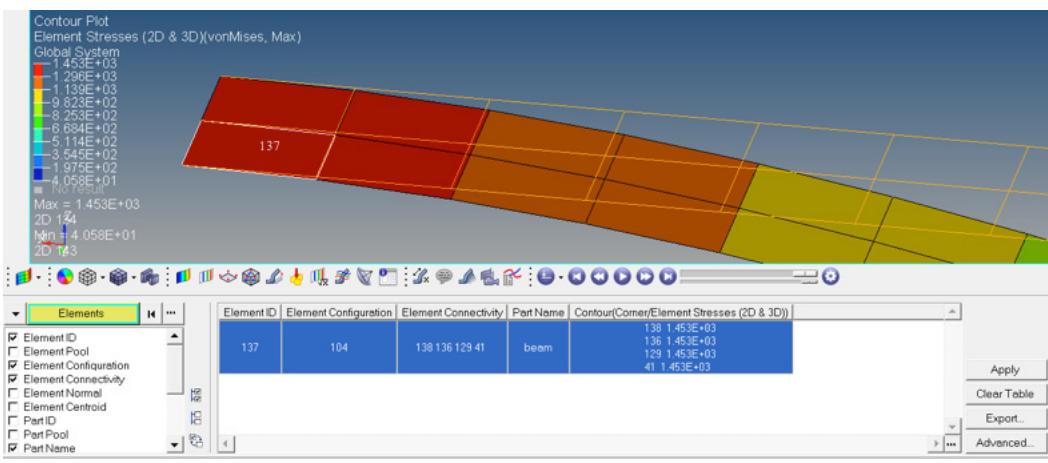


As well as another jump by about 100 MPa from element 136 to element 137.

Activating the Use corner data indicates that the stress results are pretty much the same as without the bilinear averaging method!



The **Query** panel lists identical stress values at each of the 4 nodes 1453 MPa (from the same element).



In other words, bilinear averaging doesn't affect the stress results in this case.

Why?

Looking at an element from the top view, it becomes apparent that the length of the element changes uniformly. Thus, the integration points will be stretched (to a new position) uniformly too.

Element deformation
Out-of-plane bending
on upper side



All integration points have same stress
=> Bilinear extrapolation does not "work"

In case of in-plane bending, the nodes will experience different displacements. As a consequence, the integration points will have different stress too.

Element deformation
in-plane bending



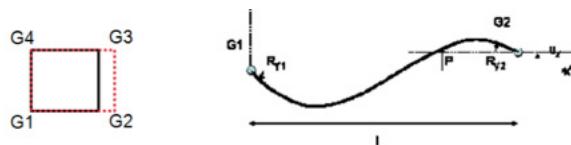
upper integration points have different stresses
=> Bilinear extrapolation does "work"

Therefore, the extrapolation option **CUBIC** may be used during the analysis.

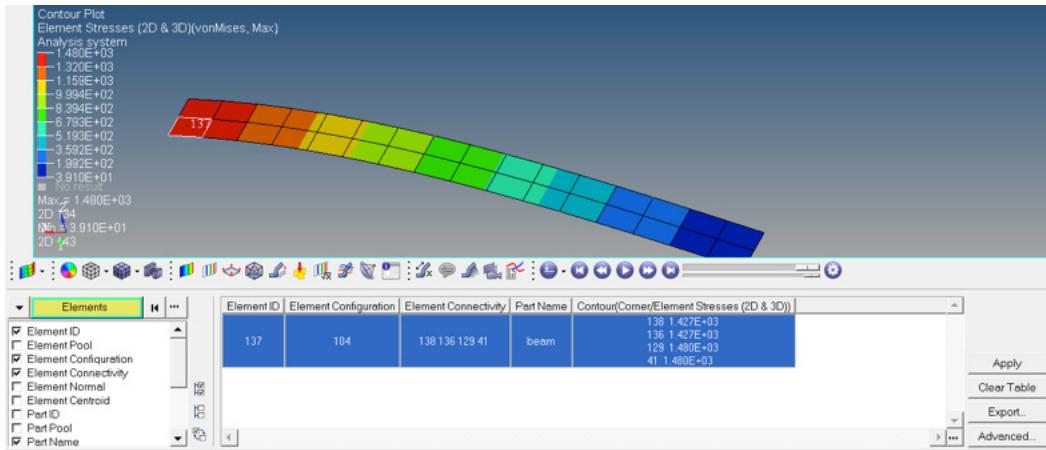
CUBIC extrapolation method can be requested with:

`STRESS(H3D,CUBIC) = YES`

This method outputs element stresses for shell elements (not available for solids) at the element center and grid points using the strain gage approach with cubic bending correction

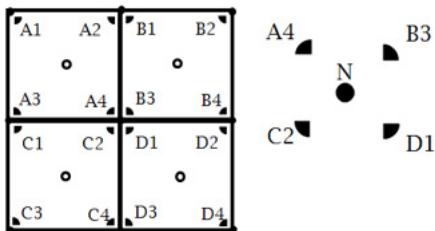


Strains G1-G2, G1-G3 and G1-G4 will be calculated from relative displacement between the nodes and used as strain-gauge rosette. In addition the bending stresses are calculated assuming a cubic displacement function like on Bernoulli beam.



Element stress based on cubic interpolation is 1480 MPa resulting in a difference between analytical and numerical result of 1.3 %.

Summary - Stress Averaging At Nodes In HyperView



Location	Method	Comment
simple	averaging of the stress invariants	Advanced should provide more accurate results
Advanced	Averaging the stress tensor and calculate invariants from this	
Difference	Max-min nodal value	Good to find areas with large stress discontinuities
Max/min	Max/min value	

Summary - Stress Locations For Shells

(recall – this information/data needs to be requested before the analysis is started)

Location	Method	Comment	Request example
Center	Integration point stresses are averaged to the center	"Correct" stresses in terms of the used FE-Model Most stable regards to element quality	Default
Corner, bilin	Bilinear extrapolation from integration point stresses to nodes	Good results for in-plane bending Cannot capture out of plane bending effects Can be very oscillatory particularly with bad element quality	STRESS(H3D,CORNER)=YES STRESS(H3D,BILIN)=YES
Cubic	Strain gauge method with cubic bending correction	Good results for out of plane bending Less oscillatory than bilin on complex stress distributions Can overestimate stresses for in-plane bending	STRESS(H3D, CUBIC)=YES
sgage	Strain gauge method	Kind of obsolete Cubic should provide more accurate results	STRESS(H3D,SGAGE)=YES

Note: Extrapolation for 2nd order shell elements is always BILINEAR

Summary - Stress Locations For Solids

Location	Method	Comment	Request example
Center	Integration point stresses are averaged to the center	"Correct" stresses in terms of the used FE-Model Most stable regards to element quality	Default
Corner, bilin, sgage, cubic	Bilinear extrapolation from integration point stresses to nodes	Extrapolation for solid elements will always be bilinear Keywords sgage and cubic are allowed to support cubic stresses for shells in same model	STRESS(H3D,CORNER)=YES STRESS(H3D,BILIN)=YES STRESS(H3D,CUBIC)=YES STRESS(H3D,SGAGE)=YES
GPS - grid point stress	Bilinear extrapolation from integration point stresses to nodes and average at node	1 value per node, whereas the others give 1 value per element Global average or average by property can be requested	GPSTRESS(H3D)=YES

Note: GPS –grid point stress can only be requested for solid elements. Here the stresses are also bilinearly extrapolated from the integration points to the nodes. However, OptiStruct then averages the stresses at each node so that only 1 stress value/result per node remain. As a consequence, the before discussed option to contour stress difference (as a measure of mesh quality) is not available any more.

Alternative Options

Shell Elements (2D)

Place Rods on the edge of shells (checking the model for free edges will create plot elements which then can be converted into rod elements). The rods (only tension/compression) are assigned the following attributes:

- Same Material as solid material
- Rod should be thin, but not too thin to jeopardise numerical stability e.g. $r=0.01$ mm

Tip: Use HyperBeam to get reasonable A and J values.

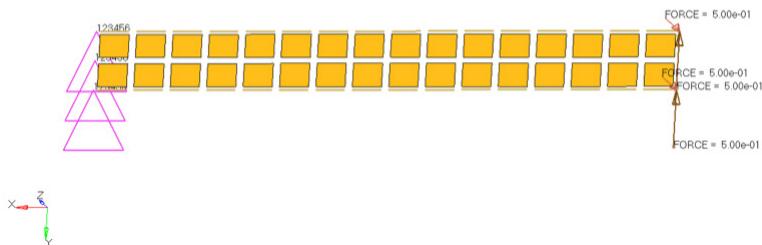
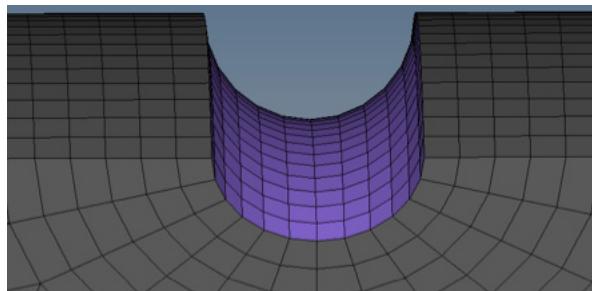


Figure: Rod elements are placed along the free edges of the shell elements

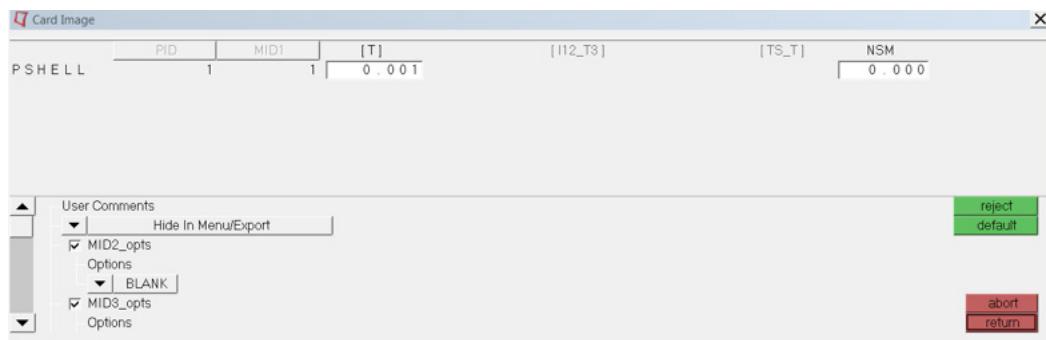
Solid Elements (3D)



Place membrane elements on the free surfaces of the solid elements (checking the model for free faces will create 2D elements

which then can be converted into membrane elements). The membrane elements (only tension/compression) are assigned the following attributes:

- Same material as solid material
- Membrane elements should be thin, but not too thin to jeopardise numerical stability
- Remove transverse shear and bending stiffness by leaving MID2 and MID 3 blank as shown in the image below (set MID2_opts (bending) and MID3_opt (transverse shear) to BLANK)



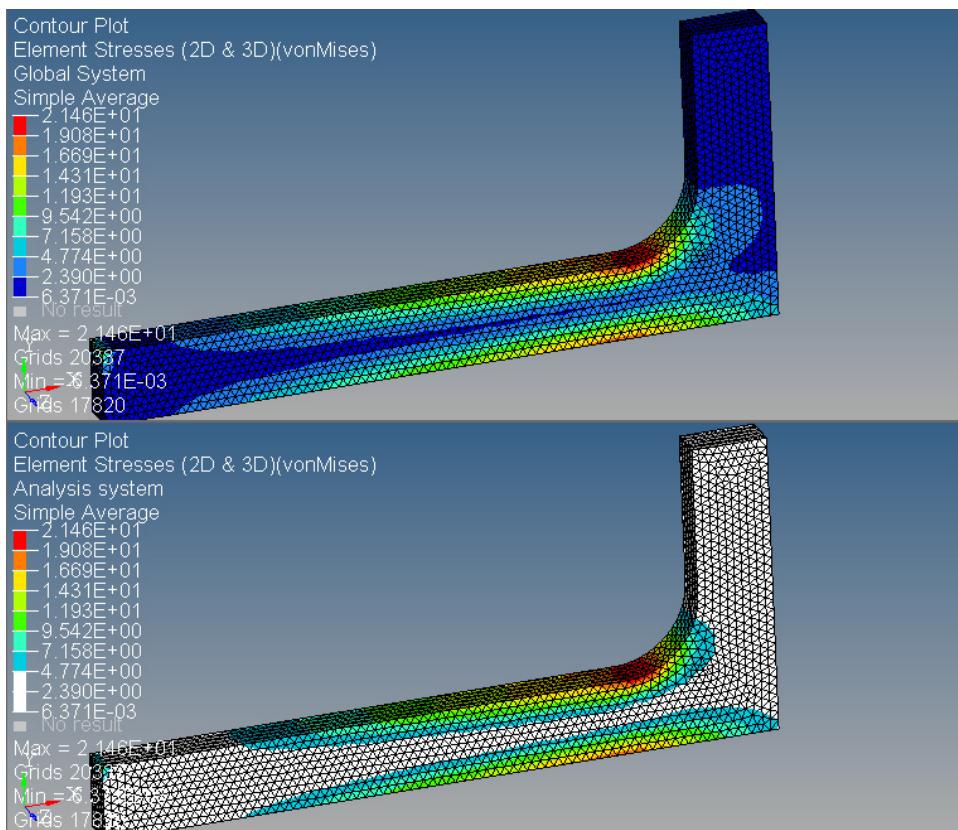
Final Remarks

- Default stress output location is at the element center, which is the “correct” stress in terms of the FE-Model.
- Usually the highest stresses of the physical part are at the surface/edge of the part due to bending effects (not true for shear dominant load).
- This can be evaluated with stress extrapolation, but there is no guarantee that the extrapolation reflects the physical behavior of the part, so it has to be used with caution.
- Alternatively, membrane/rod elements can be used to measure the stresses at the surface/edge.
- A free recorded Webinar can be viewed at: http://www.altairhyperworks.de/ResLibMP4PopUp.aspx?file_id=1711

20.5 Special Tricks For Post-Processing

1. How To Avoid Wasting Printer Ink:

A colored plot consumes a lot of ink which is very costly. In the stress contour plot, the color blue (low stresses) consumes a lot of ink unnecessarily. Wastage of ink could be avoided by using the color white in the color bar as shown below.



2. Adjusting The Scale Of The Color Bar

It may be helpful to reduce the number of colors and to adjust the legend scale. In this way all elements above a certain stress threshold (e.g. yield strength) would be displayed in red.

Elements with stresses below this threshold would then be depicted in a different color, which makes it easier to distinguish between "safe" and "failed" areas.

3. Linear Superposition Of Results

Say the results for two individual load cases F_x and F_y are already available. Now we need the result for a combined load case ($F_x + F_y$). The regular way to achieve this is to run the analysis by creating new combined load case (F_x and F_y applied together).

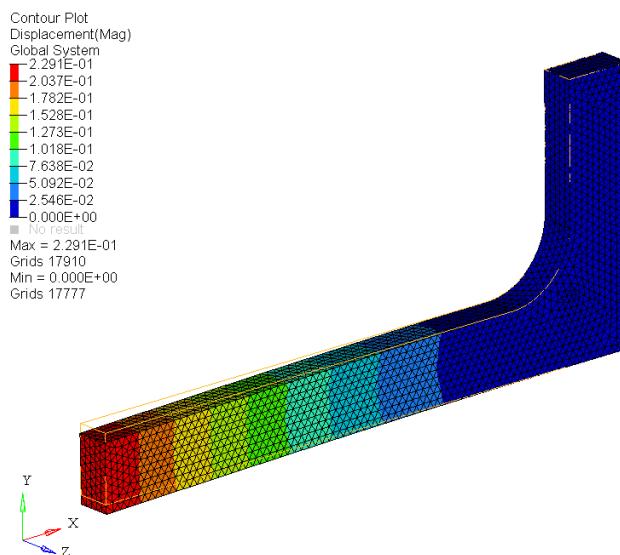
For linear static analysis, the results could be obtained even without running the analysis via (superposition of individual results):

$$\text{Result for combined load (} F_x + F_y \text{)} = \text{Result for } F_x + \text{Result for } F_y$$

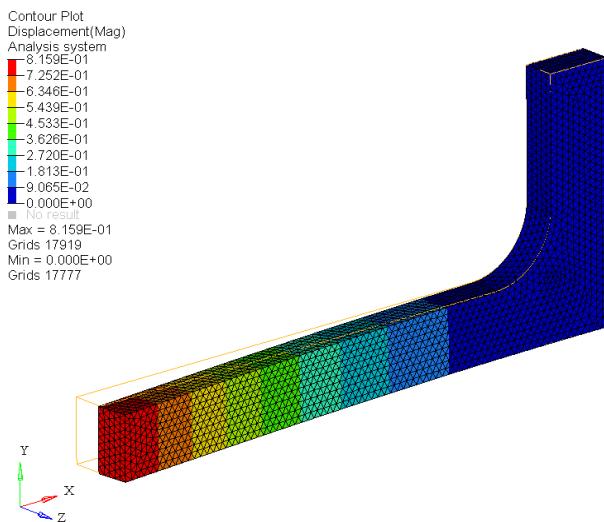
The individual load results could also be combined with an appropriate scale factor like

$$3F_x + F_y \text{ or } 2F_x - 0.5F_y.$$

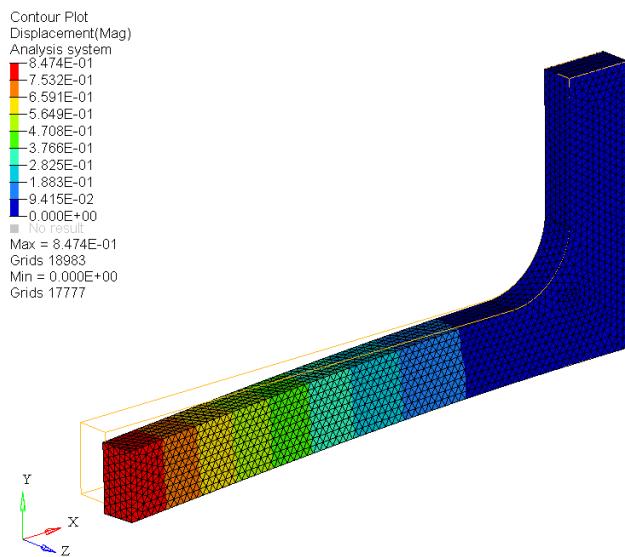
Is there any advantage in solving load cases individually when actually results are required for a combination of loads. Yes! The advantage of solving all the load cases individually is that we come to know how the individual load cases are contributing to the combined stress. This helps in subsequent corrective action for stress reduction.



Loadstep 1: vertical force. Displacements scaled by factor 30.

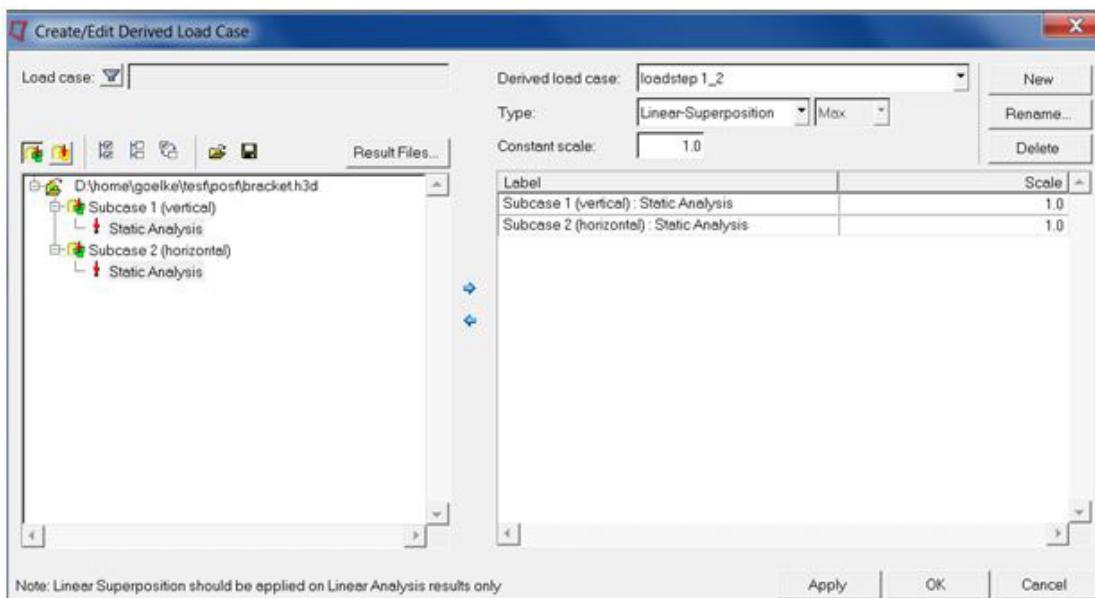


Loadstep 2: Displacements scaled by factor 30.



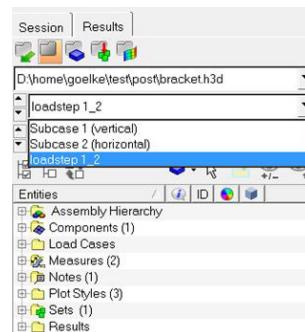
Superposition of loadstep 1 and loadstep 2. Displacements scaled by factor 30.

In HyperView loadsteps can be combined by selecting Results > Create > Derived Load Steps from the menu bar. Then select the loadsteps of interest and set Type to Linear Superposition.



Note: Linear Superposition should be applied on Linear Analysis results only.

In HyperView the superpositioned loadstep is directly accessible from the **Results Browser**.



4. Scaling Of Results:

For linear static analysis, stress is directly proportional to the force.

Therefore, when the force is doubled, the stress is also doubled. Some CAE engineers prefer running the analysis with a unit load and then specifying the appropriate scale factor to get the desired results in post processing.

5. JPEG / BMP / TIFF Format Result Files And High Quality Printouts: Common post-processors provide special provisions for stress and displacement contour plots in jpeg, bmp, tiff or postscript format files. Another simple way to achieve this is to use the Print screen command available on the keyboard.



These panels/options allow you to save your screen, panel, user defined area or even just icons as a file or directly into the clipboard. Very convenient and very helpful.

Additionally, while running OptiStruct / Radioss a report is automatically created. The report (html format) is located in your working directory (i.e. where the analysis file & results files are stored).

RADIOSS 11.0 Report

Problem submitted Tue Feb 07 20:33:24 2012
Input file D:\home\goelke\test\post\bracket.fem

Problem summary

- Problem parameters: D:\home\goelke\test\post\bracket.fem
- Finite element model: D:\home\goelke\test\post\bracket.fem
- Output files prefix: bracket

Finite element model information

- Number of nodes: 5110
- Number of elements: 20525
- Number of degrees of freedom: 14694
- Number of non-zero stiffness terms: 277095

Elements
Number of TETRA elements: 20525

Loads and boundaries
Number of FORCE sets: 2
Number of SPC sets: 1

Materials and properties
Number of PSOLID cards: 1
Number of MAT1 cards: 1

Subcases & loadcases information

Static subcases

Subcase ID	SPC ID	Force ID	Weight
1	1	2	1.00
2	1	3	1.00

Results summary

[Click here](#) to view the results with HyperViewPlayer. This feature requires a modern browser with Javascript enabled, as well as the HyperViewPlayer plugin installed.

Subcase 1 - vertical

- Maximum displacement is 0.229 at grid 17910.
- Maximum 3-D element stress is 24.9 in element 132536.

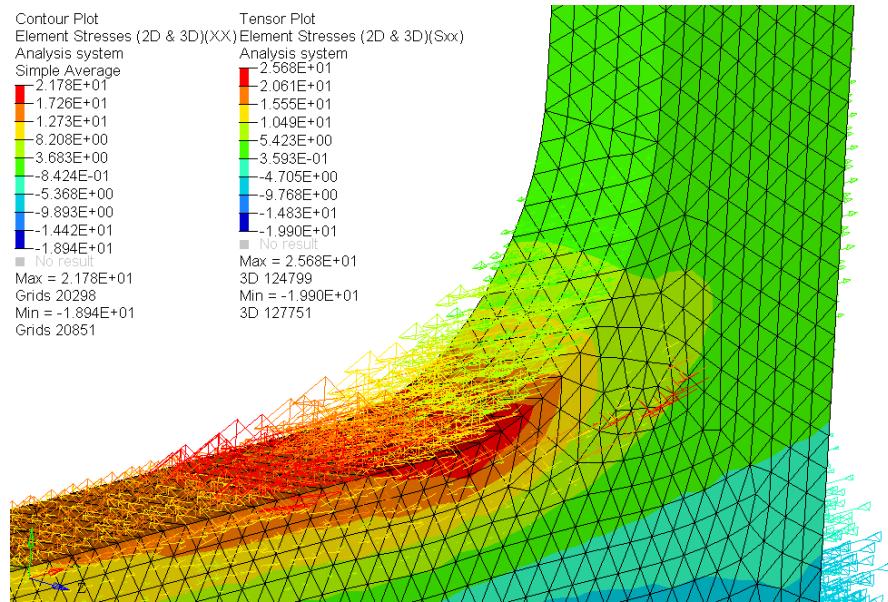
Subcase 2 - horizontal

- Maximum displacement is 0.816 at grid 17919.
- Maximum 3-D element stress is 37.4 in element 127745.

© 1996-2011 Altair Engineering, Inc.

6. Directional Stress, Vector Plot Like xx, yy etc.

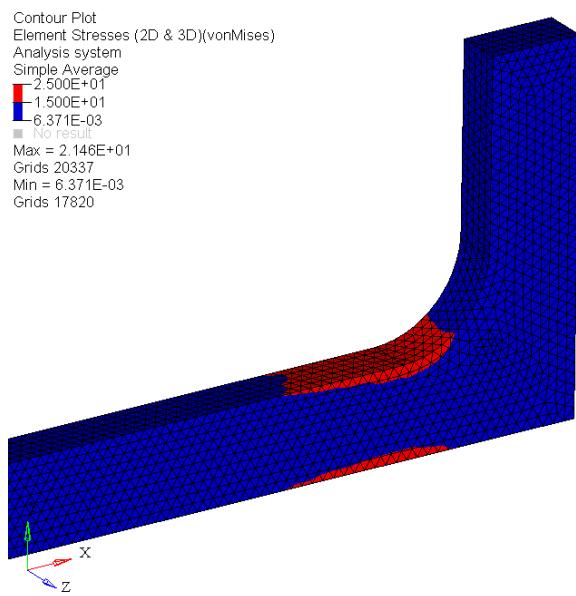
To know the direction and nature of stress (tension or compression) and for comparing CAE and strain gauge results, vector plots are recommended.



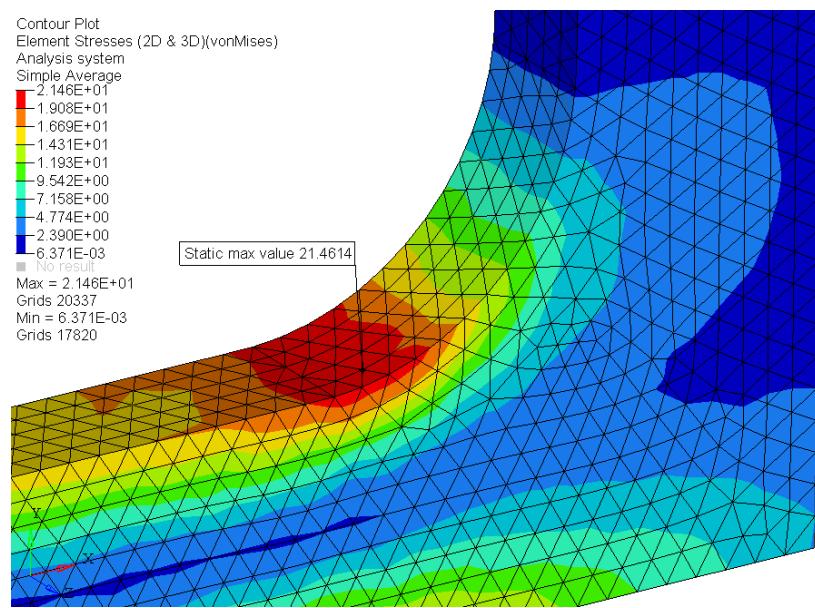
Superposition of normal stress (x-direction) contour and vector plot (outward pointing arrows indicated tension, inward oriented arrows compression)

7. Two Color Representation

The color red represent stress above the yield strength (i.e. failure) and blue shows safe areas (values are conceptual).



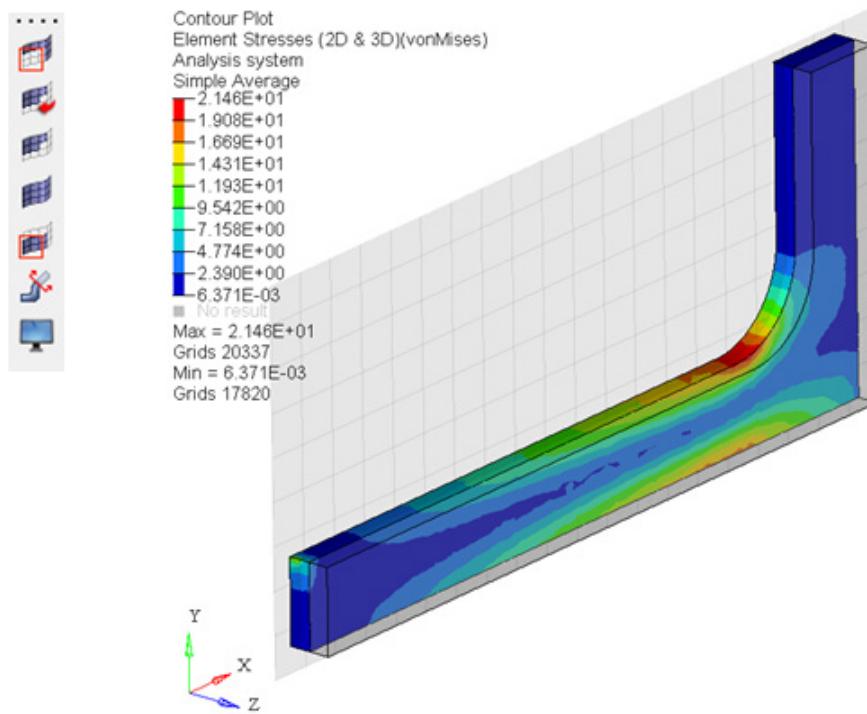
8. Always Report Maximum Stress Location



9. Stress Distribution Across Cutting Planes



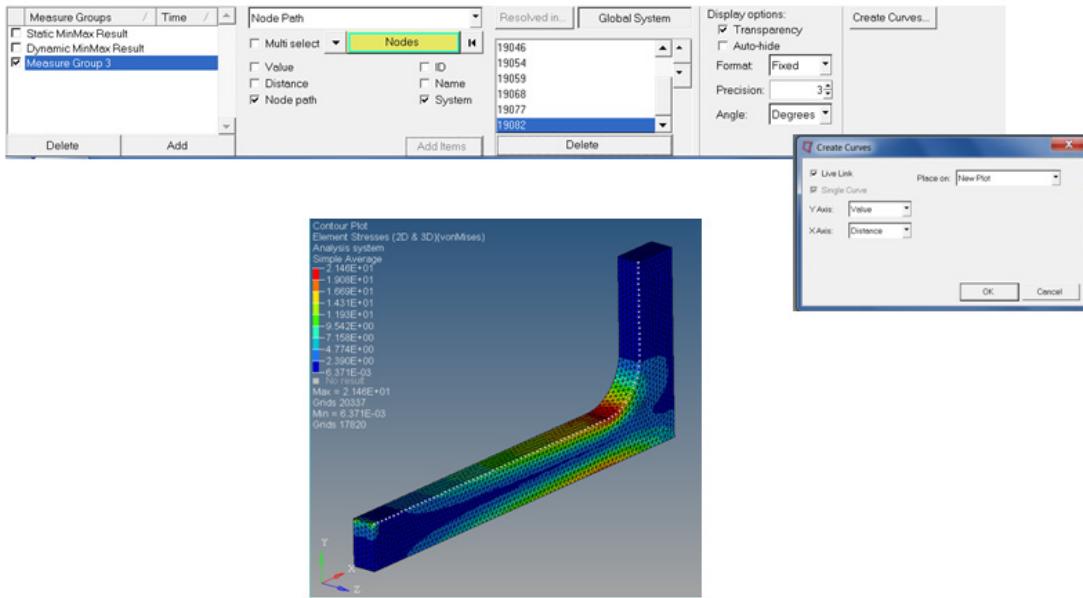
Useful in particular for 3-D elements. Just activate the corresponding icon in the side bar and define the orientation of the clipping (section) plane.



10. Graphical Display

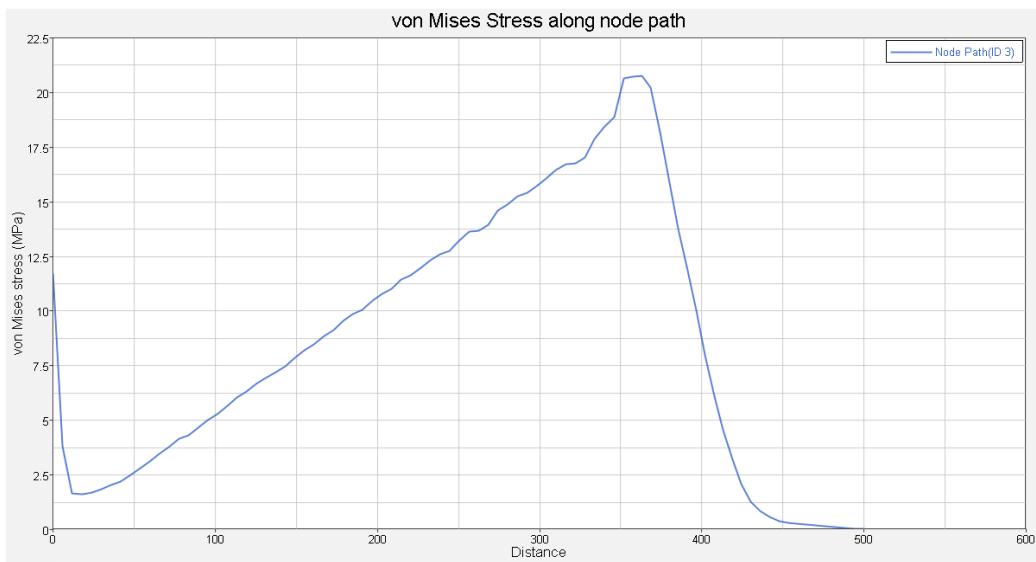


To create a graph from within HyperView activate the **Measures** panel and add a new Measure (Measure title can be renamed, of course). Depending on the kind of information you may specify “Node Path”, then select the corresponding nodes. Eventually, click on “Create Curves ...”.



The highlighted nodes represent the selected node path along which the stresses will be plotted in a graph.

Naturally, the graph layout can be adjusted according to your needs.



11. Top And Bottom Stresses For 2D Shell Elements

For 2D elements, both the top and bottom side (some postprocessors term it as Z1, Z2) stresses should be viewed. Cracks originate from the tensile side. VonMises stresses are always positive and it's not possible to know whether the displayed stresses are tensile or compressive. Directional stresses like xx, yy, or zz, using the vector plot option, could confirm the nature of the stress.

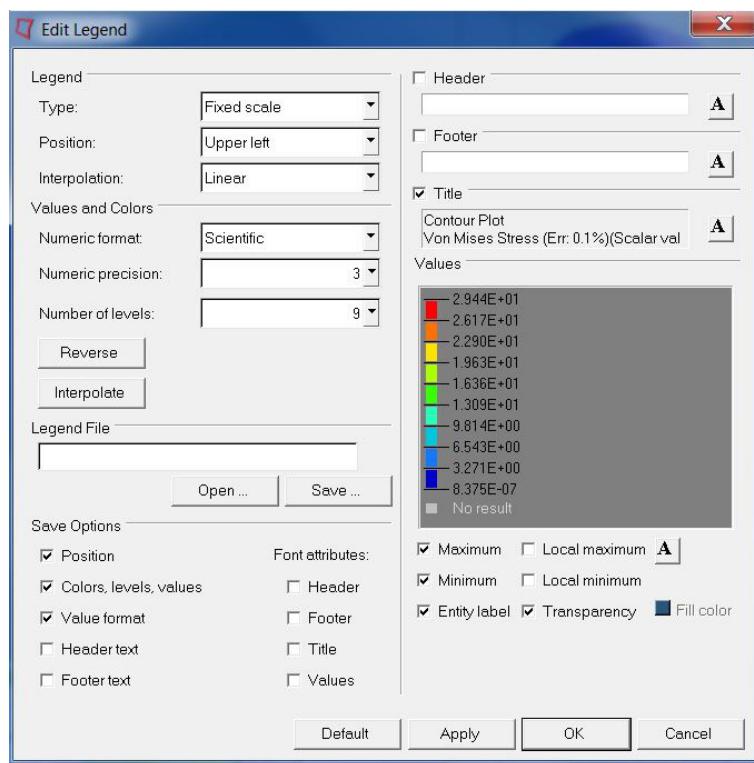
12. Result Mapping

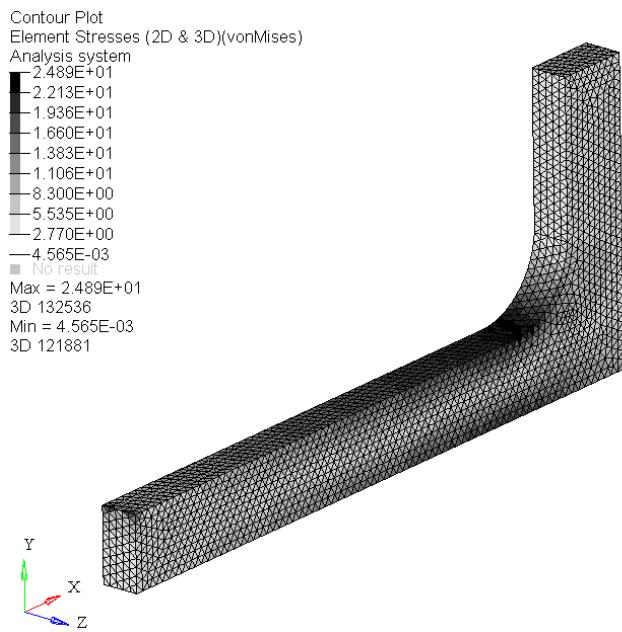
Residual / process stress consideration: Some software like Deform, HyperForm, etc. calculate the process or residual stresses. These stresses could be mapped to a FE model using a special option available in postprocessors. Consideration of these stresses during structural, crash, or fatigue analysis leads to better accuracy and correlation between the FEA and experimental results. Animate results (even of a static analysis)

13. How To Plot When Only A Black And White Printer Is Available?

This question might sound funny the first time reading it, but this is the reality. Since color plots are very costly and there is a possibility of misuse, in many organizations permission is necessary for making color plots. But there is no restriction on black and white prints i.e. contour plots are based on different gray colors.

In practice, the legend color are edited. A start color (white) and a end color (black) is defined, then the intermediate colors are automatically determined through "Interpolate".





20.6 Interpretation Of Results And Design Modifications

Based on stress and displacement contours

First step is to observe the locations of the maximum displacement and maximum stress. Also, the locations of minimum displacement and minimum stress should be found.

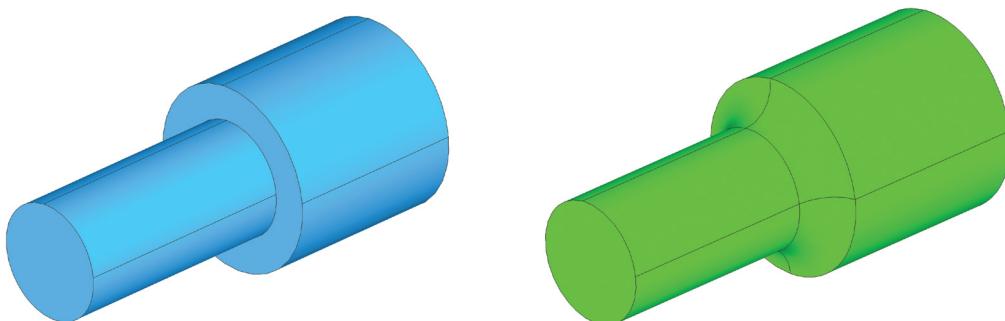
A general rule to reduce the stress or displacement is to provide a connection (using ribs or stiffeners) between the maximum stress or displacement location and the minimum stress or displacement location.

Using Strain Energy Plot For Modification

One of the tools used for suggesting modifications is based on the elemental strain energy plot. Strain energy is elastic energy stored in the element, defined as $\frac{1}{2}$ stress*strain*volume. Modifications in the region of the maximum strain energy (such as increasing the stiffness, addition of material, etc.) is recommended while low strain energy areas are good for providing bolted or welded joints or material removal from an optimization point of view.

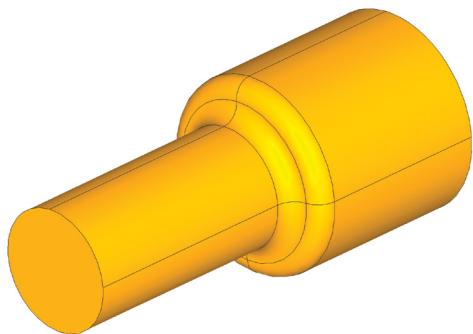
Failure Is At Sharp Corner

Suggest radius and smoothening changes

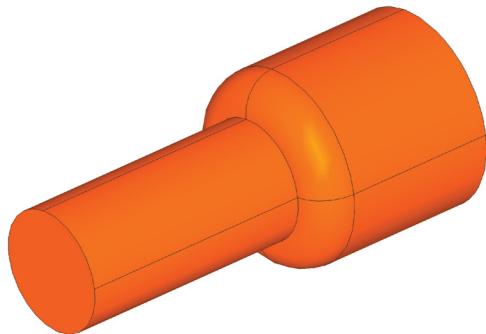


Shaft with sharp corners failing

Recommended (priority 1)



Recommended (priority 2)

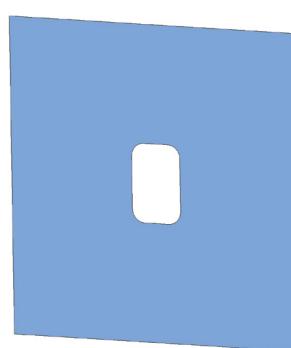


Not recommended

Rounded Corners Are Recommended Instead Of Sharp

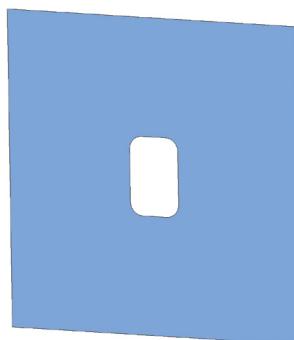


Original

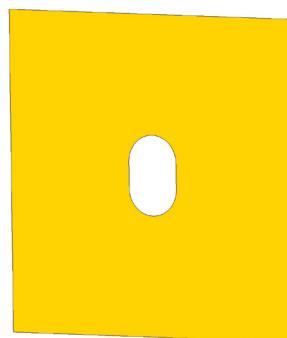


Recommended

If Failure Is At Radius – Increase Radius

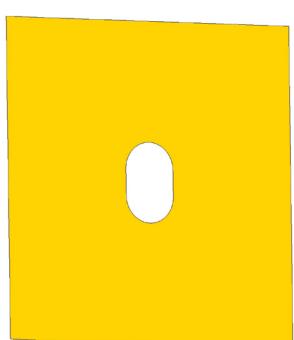


Original

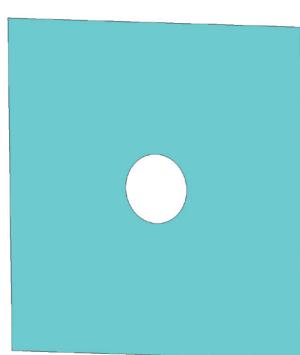


Recommended

If Failure Is At Hole Say Elliptical - Rotate It By 90 Or Try With Circular Shape Of The Hole

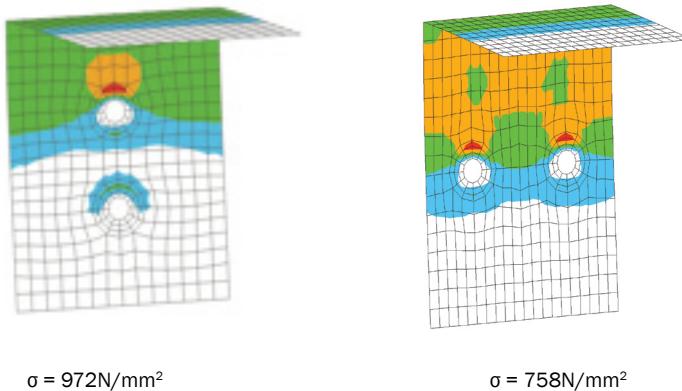


Original



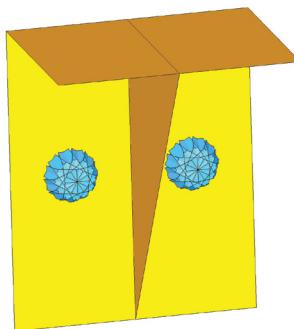
Recommended

Load Is Not Distributed Among Bolts – Rearrange Bolt Position



In the above image, the top bolt is taking most of the load while the bottom one is not effective from a design point of view. Rearrangement of the bolts along the horizontal axis would reduce the stresses without any additional cost.

If The Object Is Flexible (Large Deformation) - Add Ribs / Stiffeners



Introduction of a rib or stiffener reduces the stress as well as the displacement significantly.

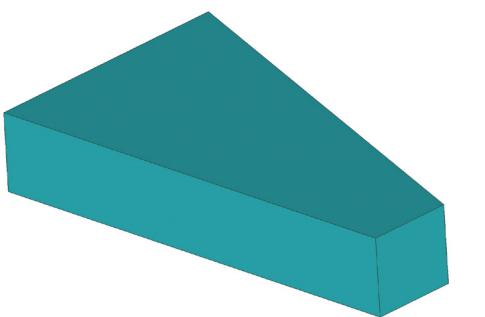
Additional Support / Fixing Point



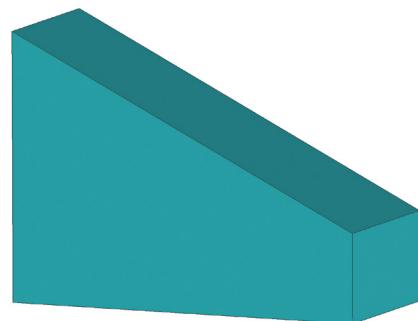
Consider a simply supported shaft (say it is located in a gear box or clutch housing). The addition of an extra support (bearing) would reduce the stress. Also, it results in a higher natural frequency and might help in the reduction of noise.

Increase Area Moment Of Inertia

To reduce the stress and displacement. Sometimes just reorientating the cross section also work very well. For example :



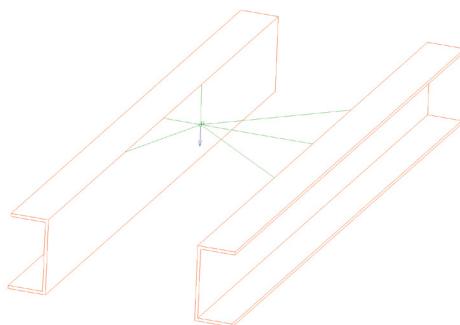
Original



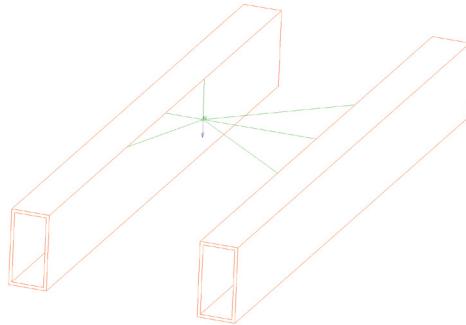
Recommended

Selection Of Appropriate Cross Section

A closed section (rectangular) is generally recommended over a C- section. In particular when the loading is not symmetric. The selection depends on the design concept. For example, in civil engineering, the thin-walled sections are very common because of the ease of use.



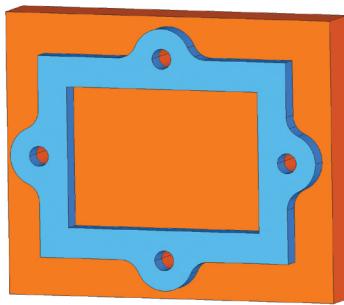
Original



Recommended

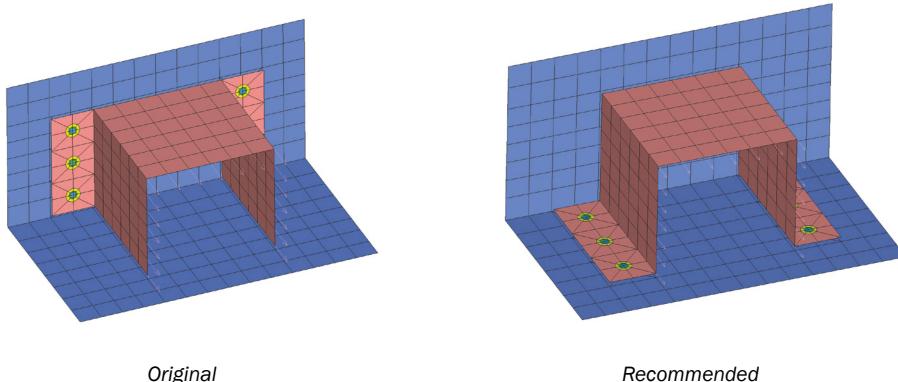
Increasing Load Transfer (Contact) Area

Two casting parts in contact – due to a surface finish, the area of the contact is small and results in less life.



Machining of contact surfaces in the above casting parts prior to assembly would give a better fatigue life (due to increased contact area).

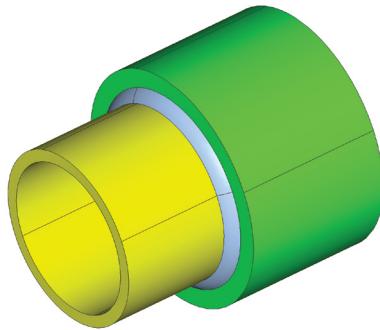
Spot Weld Is Stronger In Shear- Reorient Welding In The Case Where Failure Is Reported



Spot welds are stronger in shear and weak in normal (tension, compression, bending) loading. Many times just rearranging the spots (say orientation of the spot is changed by 90 degree) works well and solves the problem.

Arc Weld Stress Could Be Reduced By Increasing Contact Area Between Two Joining Parts

The following two cylinders (yellow and green) are welded at the inner and outer side.



A slight press fit (or interference fit) between the two cylinders prior to welding can increase the life substantially (without the press fit, all the load would be transferred to the weld, while press fitting will increase the load transfer area and reduce the stresses at the weld).

Avoiding Arc Welding For Sheet Metal Parts Having A Thickness < 0.8 mm.

Spot welds should be used for such cases.

Introduction Of Favorable Residual Stress

Shot pinning, nitriding, flame and induction hardening, and cold rolling induce favorable residual stress and increases the life of the component.

Keeping The Design Simple And Restricting The Number Of Parts

The general rule of thumb is that a single piece part has a higher strength than the same geometry produced by combining many small parts than one bolted and welded joints. When the product is in the design phase, a one piece suggestion is recommended. But when the product is already in the field and it's a case of failure analysis, then patching up the work by the addition of ribs or stiffeners is recommended. A one piece design and changes for such a situation would mean throwing away existing dies, jigs, and fixtures, and thus a major tooling changes which is quite costly and time consuming too. CAE engineer should adapt a flexible approach and suggest the best feasible design modification, while taking in to account the cost, strength, and manufacturing ease.

Increase Thickness

Increasing the thickness should be the last option as it is costly as well new dies are required (if it is increased beyond the capacity of existing ones).

High Strength Material For Highly Stressed Component

The higher strength material could be an option when failure occurs. Changing the material will not require rerunning the analysis for stress (in case of linear static analysis) and a decision could be taken just by comparing the maximum stress value with the yield / ultimate strength of the material. Higher strength materials are costly and should be considered only when other options are not giving satisfactory results.

Low Strength Material For An Over Designed Components

When the magnitude of stress is far below acceptable limit, a low strength material could be suggested. For example existing material is cold rolled (high strength) and reported stress magnitude is well below yield stress (and endurance limit), hot rolled material has low strength as well as low cost and could be suggested for cost reduction.

20.7 CAE Reports

Documentation and proper data storage or backup after the completion of the project is essential and should not be avoided.

After completion of the project, the following documentation is recommended.

- A) 2 hard copies of the report (precise and to the point short report)
- B) Power point presentation
- C) Animation files

A) Hard Copy:

CAE report should include :

Page 1: Title Or Front Page Of Report: The title of the project, a nice small figure of the component, a report number, date of submission, name of customer, name of analyst, company name, address, contact details of the analyst.

Page 2: Summary (Maximum 1 Page) Of The Project: A summary should be written in simple language (avoiding core technical terms) clearly stating the objective of the analysis, conclusion and recommendations. It is meant for managers, decision makers and team members involved in the project who either do not have sufficient time to go through the complete report or are not familiar with FEA terminology.

Signature Of CAE Engineer: The CAE engineer should sign the report, preferably on the page of the summary. This shows that he/she is taking responsibility of whatever has been written in the report.

Page 3 And Onwards: FEA technical report

- i. Aim / scope of project
- ii. About component / assembly, basic design details , functionality etc.
- iii. Methodology or strategy of analysis
- iv. Mesh details, quality checks, connection details with appropriate plots
- v. Material properties
- vi. Boundary condition details with separate figure corresponding to each load case
- vii. Tabular results for various load cases, various iterations/modifications etc. (preferably in *.xls sheet)

- viii. Result plots
- ix. Figures clearly showing recommendations/suggestions/modification to the original design.

B) Power Point Presentation

Nicely prepared power point presentation, briefly covering all of the above points should also be submitted along with the hard copies and animation files. Power point presentations are required for project presentation by the CAE engineer, group leader, and managers.

C) Animation Files

Animation or movie files are very useful in understanding the deformation as well the variation of stress. Also it could be opened easily on any PC without the necessity of having a FEA software license at the customer end.

20.8 Post-Processing Tutorials and Videos

Recommended Tutorials

These tutorials are part of the HyperWorks Help Documentation

Animation

- HV-1000: Loading Model Files
- HV-1010: Using the Animation Controls

Visibility And View Controls

- HV-2000: Controlling the Model View
- HV-2020: Using the Results Browser
- HV-2030: Masking Elements

Result Data Analysis

- HV-3000: Contouring Results
- HV-3010: Viewing Deformed Shapes
- HV-3020: Viewing Iso Values
- HV-3030: Editing Legends
- HV-3040: Viewing Vector Plots
- HV-3050: Viewing Tensor Plots
- HV-3060: Transforming and Averaging Stresses
- HV-3070: Creating Derived Loadsteps
- HV-3080: Creating Linear Superposition Loadsteps
- HV-3090: Creating Envelope Loadsteps
- HV-3095: Generating CFD Plots/Streamlines
- HV-3099: Creating Derived Results

Querying

- HV-4000: Querying Results
- HV-4010: Performing Advanced Queries

Working With The Model

- HV-5000: Creating Section Cuts
- HV-5010: Using Exploded View

Working With The Model (see HyperWorks Help Documentation)

- HV-5020: Tracking Entities during Animation
- HV-5030: Tracing Nodes and Components during Animation

Annotations

- HV-6000: Creating Measures for an FEA Model
- HV-6010: Creating Notes
- HV-6020: Using Video Overlay

Recommended Videos

- Self paced HyperView training class (HyperView / HyperGraph 12.0)
- Result Math Overview
- HyperView – published by **Prof. J. Chessa** (Texas) on Youtube

Recommended Reading

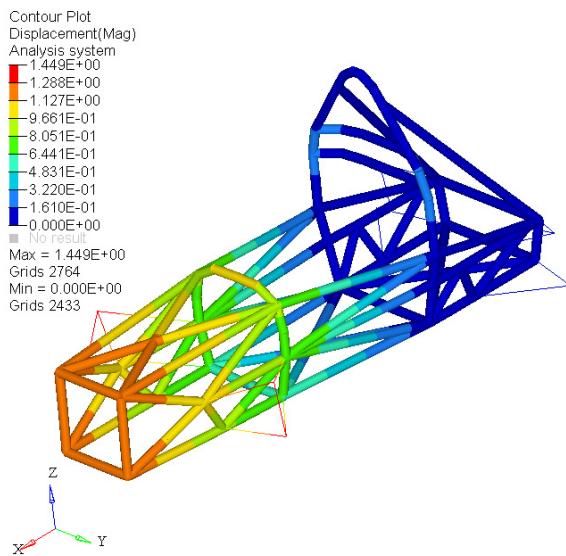
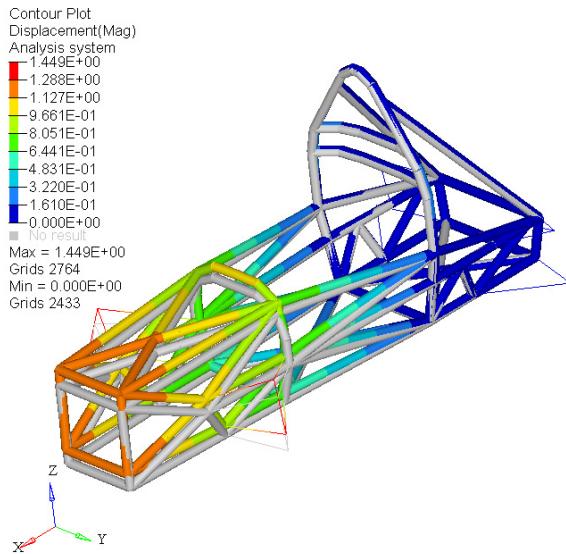
- Help Document: HyperView > Graphical User Interface > HyperView Toolbars: Results Toolbar
- Help Document: HyperView > Animation Types (HyperView)
- Help Document: HyperView > Results > Deformed Panel
- Help Document: HyperView > User's Guide > Animating Results > Contour Panel

20.9 Student Racing Car Project: Post-Processing

Finally, we are there – the FEM solver OptiStruct reports: solution is completed.

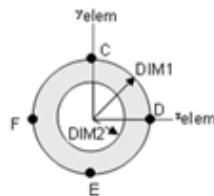
The results shown below are just “conceptual”. The detailed evaluation of the car frames performance (i.e. displacements and stresses) is deliberately left to the “reader” of this tutorial.

Bending Loadstep



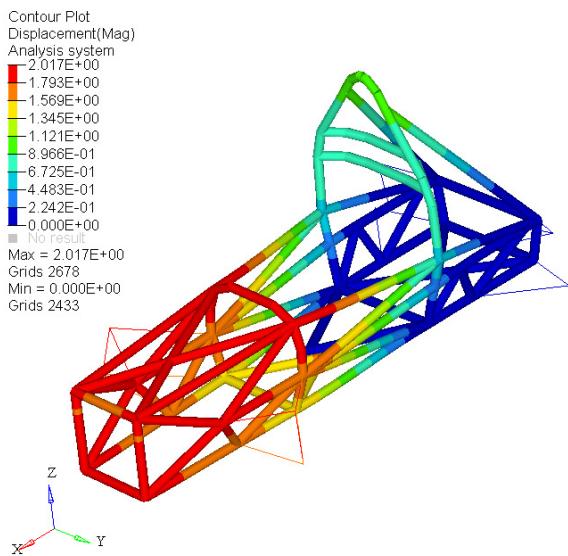
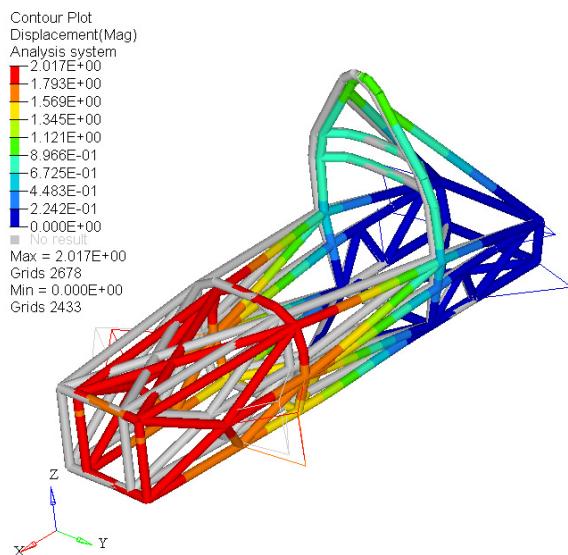
The above images show the bending loadstep contoured with the magnitude of displacements (in mm). Displacements are scaled by factor 50. Undeformed structure is displayed in gray.

In order to display element stresses, one needs to recall the definition and location of the stress recovery points of the CBAR element.



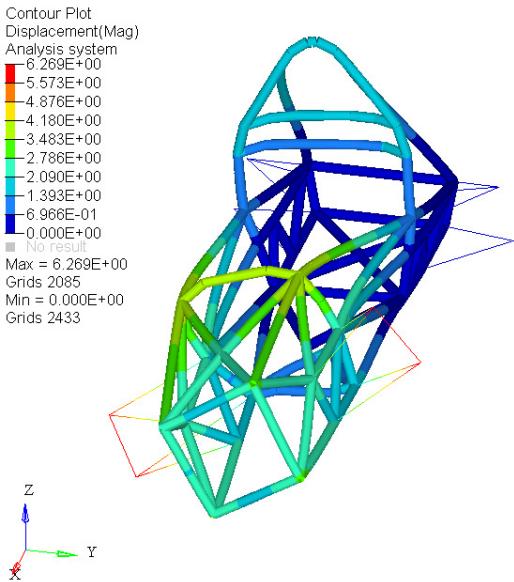
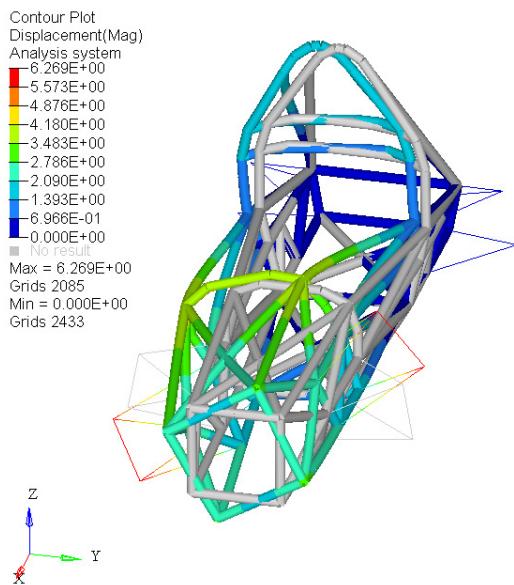
Of course, the orientation of the CBAR element now determines whether for instance recovery point C experiences tension or bending. Furthermore, it is important to realize that the stress value at the node of consideration (for instance node A), is then used to contour the entire element!

Shear Loadstep



The images above show the shear loadstep contoured with the magnitude of displacements (in mm) and scaled by a factor 50. Undeformed structure displayed in gray

Torsion Loadstep



The figures above show the torsion loadstep. The model is contoured with the magnitude of displacements (in mm) and are scaled by a factor 50.

Summary Base Design

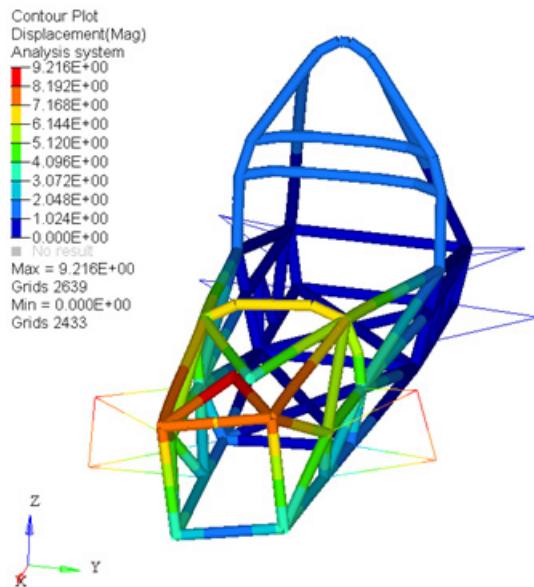
Total mass 42.1 kg

$r_o = 12.5$ mm

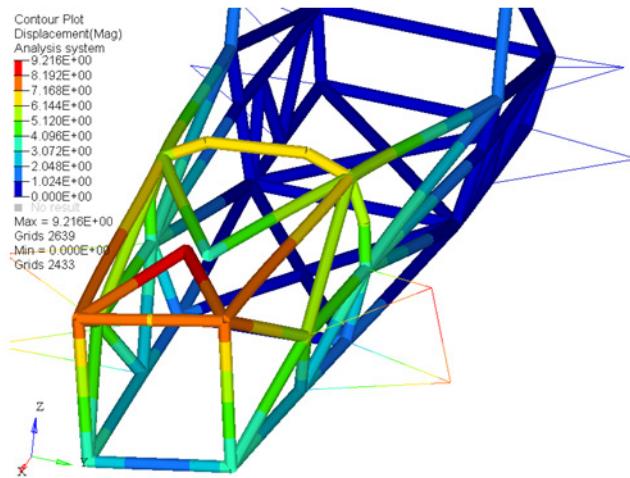
$r_i = 10.5$ mm (Note, the initial radii used in the base design are just a first guess).

Load Step	Maximum Displacement (mm)	Node
Bending	1.45	2764
Shear	2.03	2678
Torsion	6.29	2085

Quiz: Torsion Loadstep. Do You See Any Errors In Here?



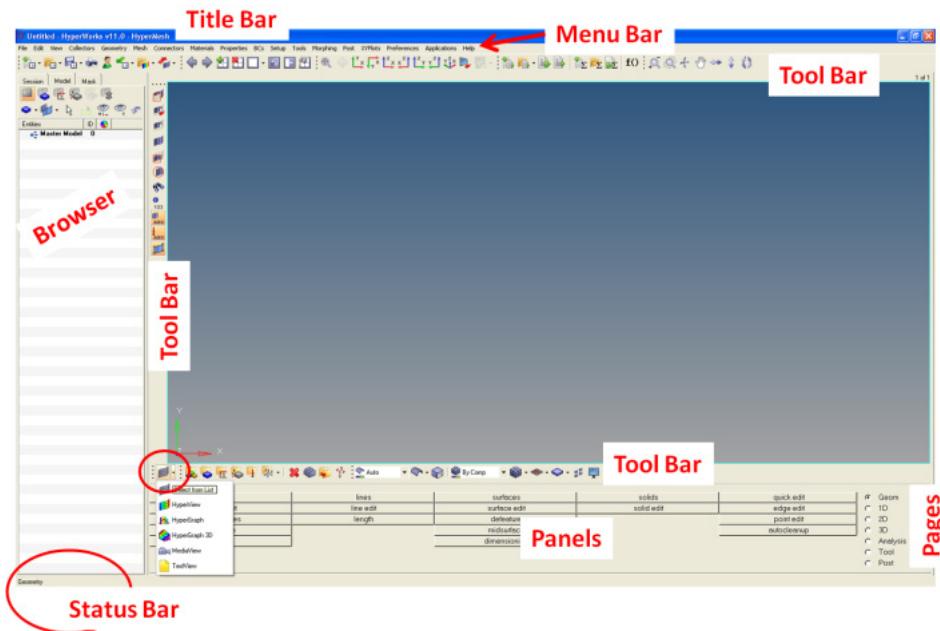
Torsion load case: Displacements scaled by a factor 10.



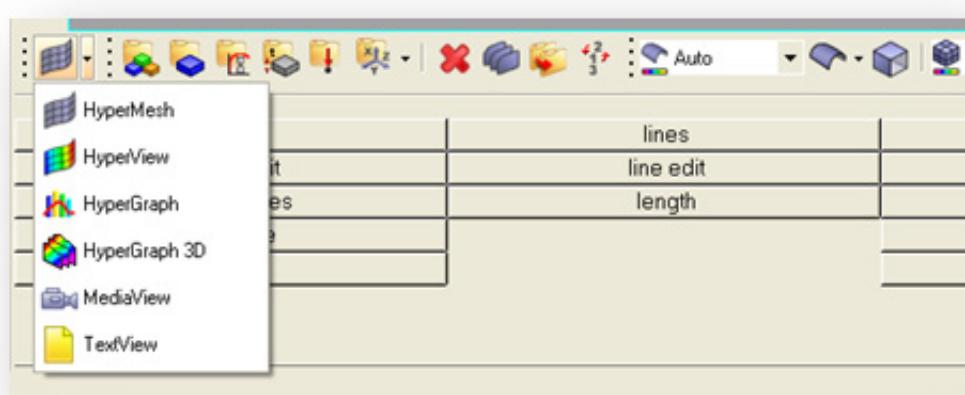
One lesson to be learned: Not every modeling mistake necessarily causes the FEM program to abort its run. In other words, it is your responsibility to understand and judge simulation results ...

21 Getting Started With HyperWorks

21.1 The HyperWorks Desktop Graphical User Interface



In the HyperWorks Desktop environment you may switch between different applications (clients) such as HyperMesh (FEM Pre-processor), HyperView, and HyperGraph (Post-processors) as shown below



In addition, the screen may be divided according to your needs, allowing you to place, for instance, a HyperMesh window next to a HyperView window.



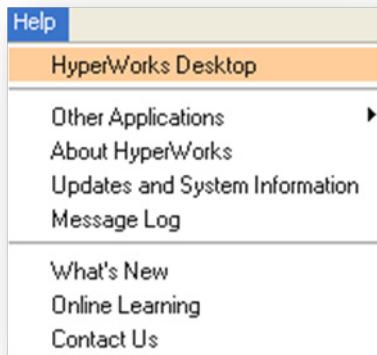
The HyperWorks Help Documentation

The HyperWorks Help documentation is part of the HyperWorks installation and/or can be viewed online (no installation needed).

Thus, the help menu may be accessed in different ways. For instance, just activate "Help" in the Menu Bar.

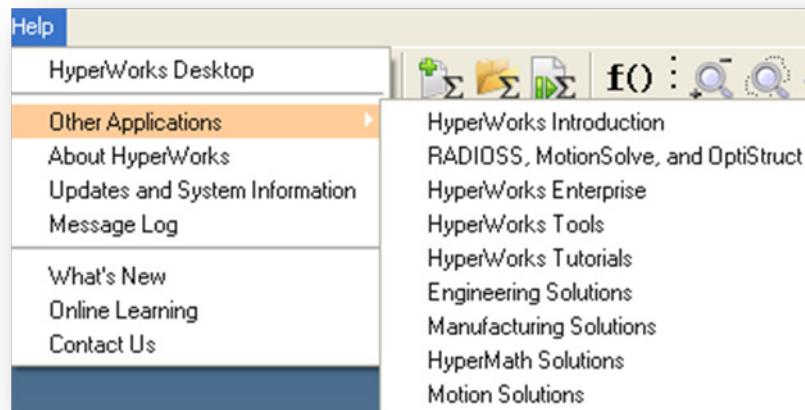


Under Help > HyperWorks Desktop, you will find answers related to "panels/functions" and working techniques.



Tip: Just press "h" on your keyboard while you're inside a panel of HyperMesh and you will be directly guided to the corresponding help entry!

Whenever you wonder about FEM solver related issues such as syntax, material definitions etc. you need to go to “Other Applications“

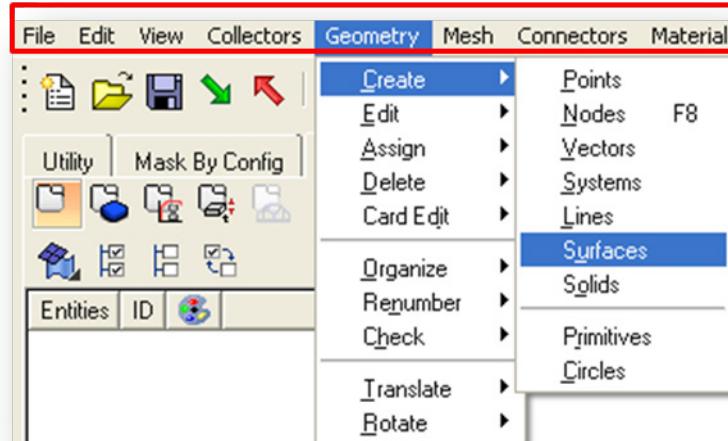


The HyperMesh “Philosophy”

Also view in the Help documentation: HyperMesh > User’s Guide > The HyperMesh Environment

Tip: Start learning HyperMesh by using the Menu Bar.

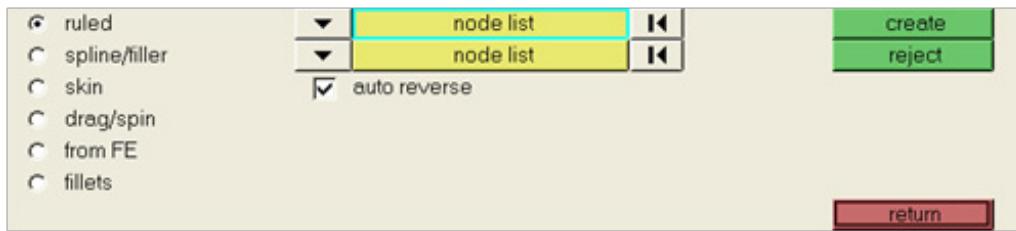
The topics/titles in the Menu Bar (e.g. Geometry, Mesh, Materials, Properties, Bc's etc.) and the pull-down menus are organized in such a way that you can build “sentences“. For instance, building a surface:



The “sentence“ would read: Geometry > Create > Surfaces

After that a panel will be displayed offering different methods on how to create the surface.

How to read panels (e.g. create surface—ruled)?



Yellow = HyperMesh expects your input here.

Blue colored frame = HyperMesh expects your input here (Note: you can jump between yellow panels by mouse clicking or by activating the tab-button).

Green=Action. In this case the surface would be build with respect to the specified lines.

Red=Close panel. No further action.

Tip: Always watch the “Status Bar” (left lower corner) for information!

At least two lines must be selected.

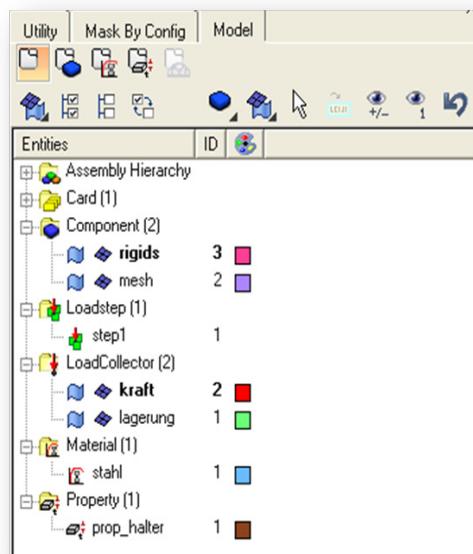
The information provided there is instrumental for your work (at least until you have a sound understanding on how HyperMesh works). It is exactly this “spot” which tells you what is happening, what is missing, what was created etc.!!

Model Browser

In HyperMesh there are many browsers. The most important one to get started with is the Model Browser.

Help document: HyperMesh > User’s Guide > Browsers > Model Browser

All entities of the FEM model such as elements, geometry, materials, properties, loads, etc. are stored in corresponding collectors:



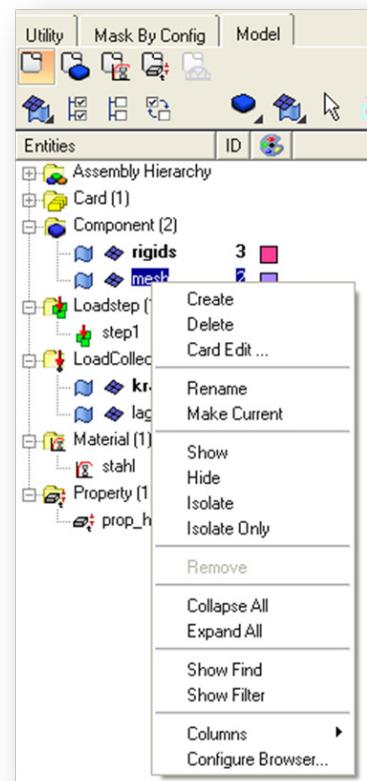
The most important collectors (at least at the beginning) are:

- component collector (comps) for elements & geometry
- material collector (mats) for material definition
- property collector (props) for properties such as thickness of shell elements
- load collector (loadcol) for loads (constraints, forces, pressure etc.)

Collectors are best created within the **Model Browser**. The **Model Browser** is a kind of “list of contents” providing information about the loaded FEM model.

In the image above, the collector written in **bold** letters (i.e. rigids) is **currently active**.

The active collector is the storage place for newly created entities belonging to this brand of collector. In this example the active component collector named “rigids” would be the storage place for all newly created elements and geometry entities (regardless whether the name of the component collector is meaningful or not).



Collectors can be made active simply with: **Make current** (right mouse button click on the corresponding component will open up a dialog window as shown in the inset on the right)

Each collector owns a number (ID) and a color. The ID is being used by the FEM solver, the color is used for visualisation purposes.

At the same time the **Model Browser** allows which FEM entities are being displayed.



Elements of the corresponding component may be displayed or non-displayed by activating – deactivating the icon.



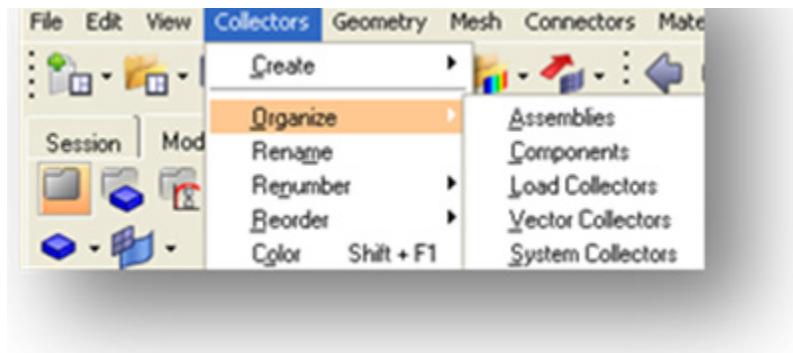
Geometry of the corresponding component may be displayed or non-displayed by activating – deactivating the icon.

Note: All entities MUST (and will be) be placed in a corresponding collector.

If you miss to create the respective collector before the entity is created, HyperMesh allocates a collector of the corresponding type automatically. This collector is named **auto**.

Don't worry— you can change it's name, color etc. at any time. It is just a right mouse click in the Model Browser.

Of course, you can also move entities from the same type of collector (e.g. component collector) to another – at any time.

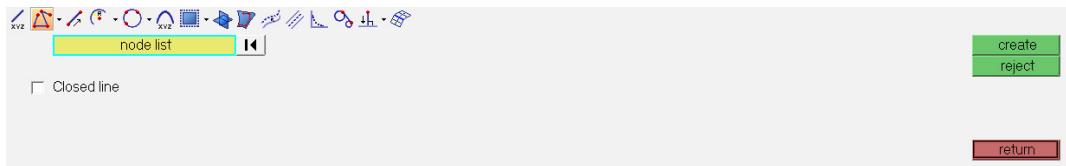


Selection Of Entities

(entities = elements, geometry etc.)

Help document: HyperMesh > User's Guide > The HyperMesh Environment > The HyperMesh User Interface > Input Controls

In order to select, for instance surfaces, it is imperative that the activated (opened) panel requests this kind of information.

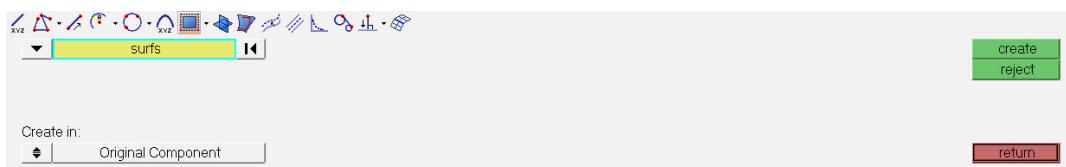


Example: The panel Geometry-Create-Lines-from nodes

In this panel you can't "read" anything about surfaces. Hence, there is no way to select/pick surfaces of the model while you are in this panel.

In contrast, in the Geometry-Create-Lines-from surf edges panel you are "asked" to provide the information from which surface edges lines should be derived. Hence, you can (actually have to) select surfaces. The respective surfaces can be selected directly via left mouse button (deselection = right mouse button).

Of course, other selection options do exist.

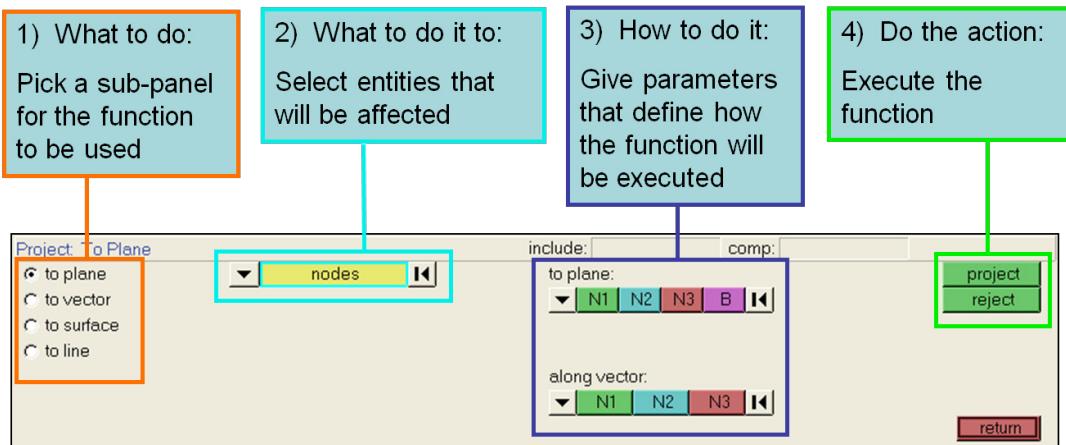


Read More:

Help document: HyperMesh > User's Guide > The HyperMesh Environment > The HyperMesh User Interface > Input Controls

Panels: General Layout

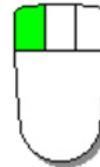
Panels often have sub-panels. These are accessed by radio buttons on the left side of the panel. Also, panels generally work from left to right. Below is an example of the **Project / to plane** sub-panel.



Mouse Buttons

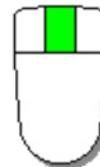
Left mouse button

- +CTRL & drag for rotate
- +CTRL & click on entity to change center of rotation
- +CTRL & click in graphics area, off entities to reset center to middle of screen



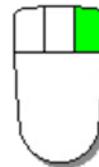
Middle mouse button

- +CTRL & drag for zoom
- +CTRL & click for fit



Right mouse button

- +CTRL & drag for pan

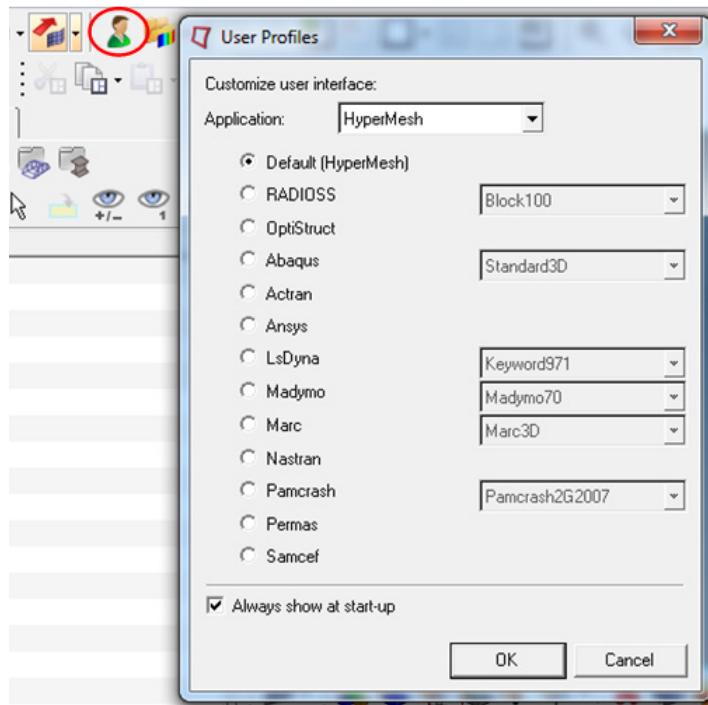


User Profile

Most likely your work will not end with meshing.

Whenever FEM solver specific information needs to be applied to the model, it is again imperative to “inform” HyperMesh about the FEM solver to be used.

This is managed through the User Profile.



Note: In the HyperWorks Student Edition not all of the above listed solvers are supported (and hence not listed).

However, in general HyperMesh allows you to build FE models for third party (non Altair) solvers as well. For instance, you can export your model in the format of Abaqus, Ansys, LS-Dyna, Marc etc.

21.2 Tutorials And Videos

Recommended Tutorials (HyperWorks_Help Documentation: HyperMesh Tutorials > Basics >)

- Getting Started with HyperMesh - HM-1000
- Opening and Saving Files - HM-1010
- Working with Panels - HM-1020
- Organizing a Model - HM-1030
- Controlling the Display - HM-1040

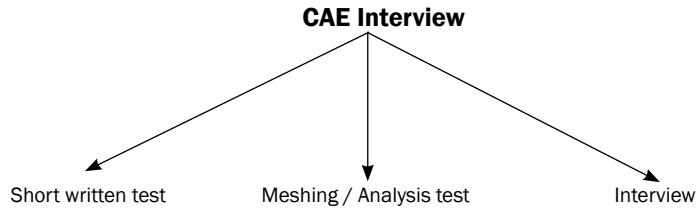
Recommended Videos

The following videos are based on HyperWorks Desktop 11.0. Nevertheless, the basics concepts shown are still valid.

- HyperWorks Desktop Framework (<http://altair-2.wistia.com/medias/r9kqmwek8n>)
- Working with “panels” (<http://altair-2.wistia.com/medias/qqpvqvqk90>)
- File import, save model (<http://altair-2.wistia.com/medias/tmmp3hl894>)
- Organize Entities (<http://altair-2.wistia.com/medias/2m355n5gnh>)
- Mask by Config Browser, Model Browser & Collectors
- Zoom, Rotate, Pan (<http://altair-2.wistia.com/medias/fievumqbpf>)
- Selection of Entities (<http://altair-2.wistia.com/medias/2vlscp3l6a>)
- User Profile in HyperMesh and Model Import (<http://altair-2.wistia.com/medias/7di95lse9d>)
- Model organization (partitioning) (<http://altair-2.wistia.com/medias/yaues86zj9>)

22 Preparing For An Interview

This chapter includes material from the book “**Practical Finite Element Analysis**”.



What Is Expected From Meshing / Analysis Test?

Software skill: Experienced CAE engineer can evaluate the performance and efficiency just by observing hand movement and the way you are operating keyboard and mouse, even from a distant place.

Submitted Meshing Job Is Checked For

1. How much work completed within allotted time
2. Methodology followed
3. Mesh flow pattern
4. Pattern around holes, geometrical discontinuities and joints
5. Mesh quality, free edges / faces, shell normal etc.
6. Whether free-free run or dummy linear static analysis carried out

Analysis Job (Ready-Made Mesh Model Is Provided To The Candidate) Is Checked For

1. Whether mesh model was checked properly before starting analysis
2. How the specified boundary conditions were applied
3. Analysis errors and warning message, and how you tackled it
4. After completion of run whether numerical accuracy checks applied (residuals, convergence test etc.)
5. Conclusions from the analysis and suggestions for design modification

FEA Theory

1. What is Finite Element Analysis.
2. What is degree of freedom (dof)? Draw following elements with their dof.
 - a. Quad 4
 - b. Tetra10
 - c. Beam element
3. What is max. principal and von Mises stress, for ductile material analysis which one would you prefer.
4. What is signed vonMises stress and absolute principal stress.
5. What is natural frequency? How many natural frequencies can any object have.
6. What is Poission's ratio. What are typical values for metal and rubber.
7. What is damping. What is the typical range of damping for metallic components.
8. What is Jacobian.
9. What is an isoparametric element.
10. What are the basic equations solved during
 - a. Linear Static Analysis
 - b. Non linear analysis
 - c. Dynamic Analysis
 - d. CFD
 - e. NVH
11. Is it possible to solve a structural analysis problem (say plate with hole subjected to tensile load) via FEM, BEM, FVM and FDM all.
12. Write two applications of each
 - i. FEM
 - ii. BEM
 - iii. FVM
 - iv. FDM
13. What is mass moment of inertia and area moment of inertia.
14. Draw engineering stress strain and true stress strain diagram for mild steel.
15. On the same diagram plot the curve for Cast Iron.
16. How would you compare performance of
 - a. Two stroke vs. Four stroke
 - b. 4 cylinder vs. 6 cylinder
 - c. Heavy vehicle and vs. light weight vehicle
17. Why a sugar cube dissolves faster in warm water than in cold.

18. Why ventilators & exhaust fans are mounted at top position on wall.
19. Why a golf ball has dimples on its face.
20. Why tornados rotates anticlockwise in Southern hemisphere & clockwise in Northern.

CAD And Geometry Clean Up

1. In what format do you receive the CAD data.
2. What are different (format) options for importing CAD data.
3. Why geometry cleanup operations are required.
4. Why mid surface is extracted.
5. For complicated geometries when mid surface operation fails how would you proceed.

Meshing

1. List 5 meshing software.
2. Why do we mesh.
3. How will you decide element type and element length for a given problem.
4. What are various quality checks, why are they needed.
5. Why % of tria elements restricted to 5%.
6. Why brick (hex) mesh when tetra mesh can fulfil the purpose and is also less time consuming.
7. Why option of linear tetra is provided in the software though it is less accurate.
8. Comment: Tria to Tetra conversion - Fixed and Floating options.
9. What is the importance of minimum element length check.
10. What is difference in structural analysis meshing, Crash analysis meshing, NVH meshing and CFD meshing.
11. Are 9 noded quadrilateral and pyramid elements possible.
12. What is plane stress element.
13. Compare tria and quad elements.
14. Why there are 2 options for tria meshing: equilateral & R-tria (Right angle tria)
15. What is the significance of shell element normal. Why it should be consistent in the model.
16. Compare Auto mesh, Batch mesh and Manual (special commands) mesh.
17. How to reduce meshing time when project time limit is very short.
18. How would you assign cross section to 1D elements.
19. What are practical applications of 1D elements.
20. What is mesh biasing.
21. How would you check mesh model submitted by a vendor.

Special Techniques

1. Difference between gap element and contact simulation.
2. How would you simulate.
 - a. Bolt and bolt pre stress
 - b. Spot weld
 - c. Arc weld
 - d. Shrink fit
 - e. Bearing
 - f. Pin joint
3. How to connect shell and solid elements.
4. How would you apply torque to a shaft modelled via solid (tetra / hex) elements.
5. What is the difference between RBE2 and RBE3 elements.

Boundary Conditions And Material Properties

1. What are the commonly used g factors for
 - a. Self weight (stand still condition)
 - b. Front or rear wheels in ditch
 - c. Braking
 - d. Cornering for vehicles.
2. What are the input material properties for
 - a. Linear static analysis
 - b. Non linear analysis
 - c. Dynamic analysis
 - d. Fatigue analysis
 - e. Crash analysis
 - f. Thermal analysis
 - g. CFD
3. What is symmetric boundary condition. Can we use it for dynamic analysis.
4. Why mass should be specified in tonne for N-mm unit system.
5. You are sitting on the chair, how would you apply boundary conditions for the same.
6. Consider a pressure vessel just kept freely on the ground (not fixed anywhere) and subjected to internal pressure, what would be the boundary conditions.
7. What boundary conditions would you apply to simulate simply supported beam.
8. What is traction, what is difference between traction and pressure.
9. How do you carry out analysis for multiple load cases, assume there are 3 load cases to be solved individually and also for the combined load.

10. How to apply sinusoidal load (dynamic/fatigue application).
11. What is PSD, in what format do you receive input data for dynamic analysis.

Analysis

1. What input data would you request to design engineer for carrying out FEA.
2. What are different solvers for
 - a. Linear static analysis
 - b. Natural frequency analysis
 - c. Forced vibration
 - d. Nonlinear analysis
 - e. Fatigue analysis
 - f. CFD
 - g. Thermal analysis
 - h. Crash analysis
 - i. NVH
3. Difference between linear and nonlinear analysis.
4. Difference between static, dynamic and fatigue analysis.
5. Limitations of linear static analysis.
6. Limitations of dynamic analysis.
7. What is stress concentration factor, if instead of steel ($E=2.1 \times 10^5 \text{ N/mm}^2$), Aluminium ($E=1.20 \times 10^5 \text{ N/mm}^2$) is used for plate with circular hole exercise, what will be SCF.
8. What is difference between h-element and p-element.
9. What are the advantages of solution restart method.
10. Is it possible to know approximate value of nonlinear stress from the corresponding results available for linear static analysis.
11. What is sub modelling.
12. What is inertia relief.
13. What are different types of nonlinearities. How would you decide when to carry out nonlinear analysis.
14. How would you carry out free-free run.
15. What is the difference between time domain and frequency domain.
16. How many modes would you consider, for dynamic analysis using modal superposition solver.
17. What is modal analysis. What are mode shapes. Is there difference between natural frequency and resonance frequency.
18. In a modal test what is coherence function, why does coherence vary with frequency.
19. What is the difference between operational mode shape and normal mode shape.
20. How to decide finite element density for vibration and acoustic analysis.
21. What type of damping models are used in vibration analysis. How to estimate damping from modal test.

22. Explain instrumentation used for modal test and sound power measurement.
23. How is sound pressure generated by vibration. How does sound pressure compare with atmospheric pressure.
24. Why we use mean square values. What is the reference for dB scale representation (sound pressure).
25. Imagine different vibration and noise problems, what appropriate modifications will you suggest. What is impedance mismatch.
26. What is High cycle and low cycle fatigue.
27. Suppose your organisation does not have a dedicated fatigue analysis software, is it possible to find approximate fatigue life using linear static results.
28. Explain steps for carrying out Fatigue Analysis.
29. Difference in implicit and explicit methods.
30. List commonly used software for NVH, CFD, Thermal and Mechanism simulation.
32. Say 3 identical rods made from steel, aluminium and brass are subjected to same force. Which one would have highest stress and displacement.
32. How do you take care of residual stresses in the analysis.
33. What could be the probable reasons for failure of any component.
34. Sketch O-grid topology for circular pipe.
35. What information do you get from streamline plot.
36. What are typical applications of crash simulation in various industries.
37. List injury criteria's for occupant safety.
38. Why in the design of a compact heat exchanger turbulent flow is preferred over laminar.

Post-Processing

1. At what location stress is reported by software.
2. Difference in Average and Unaverage stress.
3. Why stress results from a solver when viewed in different post-processors shows different values.
4. What are different options to view the results.
5. How to reduce wastage of printer ink.
6. How do you interpret the results.
7. What is linear superposition of results.
8. How to check accuracy of results.
9. Describe basic thumb rules for design modification.
10. After completion of analysis how do you report (in what format) the results to customer.

Overall, aside your technical skills and passion, your softskills (e.g. being a team player) are important for your career. It is simply a matter of efficiency whether you want to re-invent the wheel yourself, or whether you take the “wheel” and move ahead together with the team ...

We included some general recommendations on our **Academic Webiste**. SUCCESS!