Thin-wall structure simulation

Erke Wang
CADFEM GmbH
Germany
Contents

1) Introduction
2) Thin-wall structure character, locking and remedy
3) Thin-wall structure simulation with linear material - different elements
4) Thin-wall structure simulation with nonlinear material - different elements
5) Large models under Windows/XP 64-Bit
6) Summary
1. Introduction

Thin-wall structures are widely used in Aerospace, automotive, transportation, and many other industries.

Reasons: 1) material saving, 2) energy saving, 3) better performance
1. Introduction

Conventionally, because of the hardware limitation and the analysis efficiency, the thin-wall structures are modeled in FEA with shell elements.

Advantage:
- Analysis efficiency
Disadvantage:
- Engineering time

With the on-going CAD-CAE integration and the rapid performance improvement in new hardware development, more and more solid elements are used in simulating the thin-wall structures.

Advantage:
- Engineering time
Disadvantage:
- Analysis efficiency
1. Introduction

The newly developed solid-shell element technology is the another way in simulating the thin-wall structure, in which both the solid modeling efficiency and the advantage of the shell element in avoiding locking are employed.

- Involves only displacement nodal DOFs and features an eight-node brick connectivity.
- Performs well in simulating shell structures with a wide range of thickness (from extremely thin to moderate thick).
- Is compatible with 3D constitutive models and automatically accounts for thickness change.
- Performs well for both flat-plate and curved shells.
2. Thin-wall structure character

Three different ways to solve the thin-wall structure problem

- Shell
  - Shell model (discrete in $\zeta$, continuous in $\xi, \eta$)
  - Finite element discretization
    - 2d shape functions (e.g. bilinear in $\xi, \eta$)
    - Directors
    - 6 DOF per node, e.g. 4 nodes per element

- Solid
  - Finite element discretization
    - 3d shape functions (e.g. tri-linear in $\xi, \eta, \zeta$)
    - 3 DOF per node, e.g. 8 nodes per element

- Solid-Shell
  - Special element formulation:
    - e.g. assumed strain formulation
    - 3 DOF per node, e.g. 8 nodes per element
Most solid elements show the locking phenomena, when used for thin-wall structure under bending dominated loading.

Although with enhanced strain formulation and/or mid-side nodes most locking phenomena can be remedied, Solid185 and Solid186 still show transverse shear locking, when the structure is really thin, e.g. L/t = 1000 and the coarse mesh is used.

Solid185 with enhanced strain
L/t = 1000, transverse shear stress

U_{max} = 0.035, but should be 1

Solid186
L/t = 1000, transverse shear stress

U_{max} = 0.83, but should be 1
2. Thin-wall structure character

Remedy: when fine mesh is used, the transverse shear locking will not influence the global stiffness.

1. Increase the mesh density(SOLID185), a very expensive solution

2. Use mid-side nodes Solid(SOLID186), expensive solution
2. Thin-wall structure character

Remedy: Shell element and Solid-Shell element are designed to model the thin-wall structures.

3. Use Shell element (SHELL181), need mid-surface

- transverse shear stress
- $U_{\text{max}} = 1$

4. Use Solid-Shell (SOLSH190), need sweep mesh

- transverse shear stress
- $U_{\text{max}} = 1$
Software

Thin wall structure– Solution

1. Solid-Shell,
   - need a good sweep mesher

2. Shell,
   - need a good mid-surface tool
     and good shell mesher

3. Solid,
   - need a good solver
     and robust tet mesher
3. Thin-wall structure simulation

large deflection thin-wall structure simulation
- Scenario-1: linear material
- Scenario-2: nonlinear material

Large CAE models
3. Thin-wall structure simulation

It is part of a frame of an automobile. It is a cross member bracket mound for a transmission.
3. Thin-wall structure simulation

- Large deflection with linear material
  - 8-nodes Solid-Shell
  - 4-nodes Shell
  - 20-nodes Hex
  - 10-nodes Tets

- Large deflection with nonlinear material
  - 8-nodes Solid-Shell
  - 4-nodes Shell
  - 20-nodes Hex
  - 10-nodes Tets

Material

Structural Steel

- Young's Modulus: 2.e+005 MPa
- Poisson's Ratio: 0.3
- Density: 7.85e-006 kg/mm³

Structural Steel - Multilinear Isotropic Hardening

<table>
<thead>
<tr>
<th>astatic Strain mm/m</th>
<th>Stress MPa</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>1.0 x 10^-2</td>
<td>360.0</td>
</tr>
<tr>
<td>2.0 x 10^-2</td>
<td>400.0</td>
</tr>
<tr>
<td>3.0 x 10^-2</td>
<td>430.0</td>
</tr>
<tr>
<td>4.0 x 10^-2</td>
<td>450.0</td>
</tr>
<tr>
<td>5.0 x 10^-2</td>
<td>460.0</td>
</tr>
<tr>
<td>6.0 x 10^-2</td>
<td>465.0</td>
</tr>
<tr>
<td>0.1</td>
<td>469.0</td>
</tr>
<tr>
<td>0.2</td>
<td>470.0</td>
</tr>
<tr>
<td>0.5</td>
<td>470.0</td>
</tr>
</tbody>
</table>

© 2006 ANSYS, Inc.
3. Thin-wall structure simulation with linear material

- SOLSH190 Large deflection with linear material

In order to get SOLID190 mesh, the following options should be used:
1) Use Low order element
2) Use Thin Model if Possible
3) Insert command to use SOLSH190

Statistics:
- Nodes: 11,878
- Elements: 5,670
3. Thin-wall structure simulation with linear material

- SOLSH190 Large deflection with linear material

Displacement
Dmax=7.40 mm

Stress
Smax=993 MPA

Convergence monitor
3. Thin-wall structure simulation with linear material

Thin-wall structure-Shell mesh

Because of the thin character of the structure, a slightly introduced geometrical imperfection will result in an unstable solution, e.g., local buckling.

Sources of the imperfection:
1) Inaccurate geometry description due to defeaturing.
2) Improper node location due to poor mesh algorithm.
3. Thin-wall structure simulation with linear material

Thin-wall structure-Shell mesh – coarse mesh, linear analysis

Physics

Non-Uniform mesh Displacement vs. Uniform mesh Displacement

Perfect cylinder

After mesh the cylinder is imperfect

After mesh the cylinder is still perfect
3. Thin-wall structure simulation with linear material

Thin-wall structure-Shell mesh - fine mesh, linear analysis

Physics

Non-Uniform mesh vs. Uniform mesh

Displacement

The fine mesh will not necessary solve the problem

Perfect cylinder

After mesh the cylinder is imperfect

After mesh the cylinder is still perfect
3. Thin-wall structure simulation with linear material

Thin-wall structure-Shell mesh - coarse mesh, nonlinear analysis

Physics

Non-Uniform mesh vs. Uniform mesh Displacement

For nonlinear analysis, the imperfection will result in convergence difficult

Perfect cylinder

Convergence monitor

Convergence monitor

© 2006 ANSYS, Inc.
3. Thin-wall structure simulation with linear material

Thin-wall structure-Shell mesh

Default Mesher  Quad Mesher

Default Mesher  Quad Mesher
3. Thin-wall structure simulation with linear material

- **SHELL181 Large deflection with linear material**
  In order to get good shell mesh, the following options should be used:
  1) All Quad Mesh, 2) Defeaturing torelance, 3) Element size

![Mesh Details and Statistics](image)
3. Thin-wall structure simulation with linear material

- SOLID186 Large deflection with linear material

In order to get Hex mesh, the following options should be used:
1) Element order to high, 2) Use Thin Model if Possible

---

**Statistics**

<table>
<thead>
<tr>
<th>Nodes</th>
<th>41047</th>
</tr>
</thead>
<tbody>
<tr>
<td>Elements</td>
<td>5670</td>
</tr>
</tbody>
</table>
3. Thin-wall structure simulation with linear material

- SOLID187 Large deflection with linear material,

  Analytic solution vs. tetrahedron for thin-wall structure

The FE model uses one element over the thickness and the in-plane element size is calculated as following:

Element Size = Factor \times \text{Thickness}

For length/thickness ratios from 50, to 5000, the studies are done to fine the factors that meet the <2% accuracy.
3. Thin-wall structure simulation with linear material

- **SOLID187** Large deflection with linear material,

<table>
<thead>
<tr>
<th>Length/Thickness Ratio</th>
<th>50</th>
<th>100</th>
<th>500</th>
<th>1000</th>
<th>2000</th>
<th>3000</th>
<th>4000</th>
<th>5000</th>
</tr>
</thead>
<tbody>
<tr>
<td>Factor</td>
<td>3</td>
<td>5</td>
<td>12</td>
<td>17</td>
<td>25</td>
<td>30</td>
<td>30</td>
<td>20</td>
</tr>
<tr>
<td>DOFs</td>
<td>11k</td>
<td>15k</td>
<td>65k</td>
<td>127k</td>
<td>230k</td>
<td>360k</td>
<td>650k</td>
<td>2300k</td>
</tr>
</tbody>
</table>

Element Size = Factor * Thickness
3. Thin-wall structure simulation with linear material

- SOLID187 Large deflection with linear material,
  \[ L=200 \quad T=2, \quad \text{Ratio}=L/T=100, \quad \text{Factor}=5 \]
  \[ \text{Element Size} < \text{Factor} \times \text{Thickness} = 5 \times 2 = 10 \text{ mm} \]
3. Thin-wall structure simulation with linear material

Displacement
D_{max}=7.40 \text{ mm}

Displacement
D_{max}=7.39 \text{ mm}

Displacement
D_{max}=7.42 \text{ mm}

Displacement
D_{max}=7.19 \text{ mm}

Stress
S_{max}=993 \text{ MPA}

Stress
S_{max}=964 \text{ MPA}

Stress
S_{max}=1052 \text{ MPA}

Stress
S_{max}=1029 \text{ MPA}

SOLSH190

SHELL181

SOLID186

SOLID187
3. Thin-wall structure simulation with linear material

Conclusion:
- All 4 element give good answer
- SOLSH190 and SHELL181 are most efficient
- SOLID187 is the most expensive
3. Thin-wall structure simulation with nonlinear material

- **SOLSH190 Large deflection with nonlinear material**

  - **Displacement**
    - $D_{\text{max}} = 27.96$ mm
  
  - **Stress**
    - $S_{\text{max}} = 451$ MPa
  
  - **Plastic strain**
    - $E_{\text{max}} = 4.48\%$

---

**Material**

**Convergence monitor**

---

© 2006 ANSYS, Inc.

ANSYS, Inc. Proprietary
3. Thin-wall structure simulation with nonlinear material

SOLSH190
- Displacement: Dmax = 27.96 mm
- Stress: Smax = 451 MPa
- Plastic strain: Emax = 4.48%

SHELL181
- Displacement: Dmax = 26.96 mm
- Stress: Smax = 453 MPa
- Plastic strain: Emax = 4.63%

SOLID186
- Displacement: Dmax = 22.51 mm
- Stress: Smax = 422 MPa
- Plastic strain: Emax = 2.77%

SOLID187
- Displacement: Dmax = 22.27 mm
- Stress: Smax = 402 MPa
- Plastic strain: Emax = 2.41%

© 2006 ANSYS, Inc.
Let’s look at the plastic strain results:

**SOLSH190**
Plastic strain E_max = 4.48%

**SHELL181**
Plastic strain E_max = 4.63%

**SOLID186**
Plastic strain E_max = 2.77%

**SOLID187**
Plastic strain E_max = 2.41%

It’s noticed, that not only the strain values are not the same, even the plastic strain distributions are not the same.

**SOLSH190** and **SHELL181** show the same plastic strain distributions and the same value.

**SOLID186** and **SOLID187** show the same plastic strain distributions and the same value.
It is noticed, that there are two obvious different in plastic strain results between SOLSH190 and SOLID186:
1) In some regions, SOLID186 show the zero plastic strain, while SOLSH190 not.
2) SOLSH190 show much higher plastic strain as SOLID186.
3. Thin-wall structure simulation with nonlinear material

**Conclusion:**

- **SOLID186 and SOLID187 answers are incorrect:**
  When the stress beyond the yield stress, the strain distribution is not linear anymore through the thickness, SOLID186 and SOLID187 have only 3 integration points through the thickness, which is not enough to model the strain distribution through the thickness.

- **SHELL181 has 5 integration points (user can set the number) through the thickness for material nonlinearity, the answer is much more accurate.**

![Diagram showing strain distribution](image)

- **SHELL181**
  - Dmax = 26.96 mm

- **SOLID186**
  - Dmax = 22.51 mm
3. Thin-wall structure simulation with nonlinear material

- SOLSH190:
  Number of the integration points through the thickness can be defined with SECTION definition, in this case 5 points are used.

- SOLSH190: Dmax=27.96 mm
- SHELL181: Dmax=26.96 mm
- SOLID186: Dmax=22.51 mm
- SOLID187: Dmax=22.27 mm
3. Thin-wall structure simulation with nonlinear material

- Total displacement
  - SOLSH190: 27.96
  - SHELL181: 26.96
  - SOLID186: 22.51
  - SOLID187: 22.27

- Stress
  - SOLSH190: 451
  - SHELL181: 453
  - SOLID186: 422
  - SOLID187: 402

- Plastic strain
  - SOLSH190: 4.48
  - SHELL181: 4.63
  - SOLID186: 2.77
  - SOLID187: 2.41

- Elapsed time
  - SOLSH190: 1558
  - SHELL181: 1558
  - SOLID186: 8420
  - SOLID187: 16232

© 2006 ANSYS, Inc.
3. Thin-wall structure simulation with nonlinear material

-SOLID187:
Another option is to do a very fine mesh, i.e. more than one element through the thickness, this option works for all kind of geometry and the result is also good, the consequence is the FE model will be huge.

Elements: 486609
Nodes: 770966
DOFs: 3000000

SOLID190 model

<table>
<thead>
<tr>
<th>Statistics</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Nodes</td>
<td>11878</td>
</tr>
<tr>
<td>Elements</td>
<td>5670</td>
</tr>
</tbody>
</table>
3. Thin-wall structure simulation with nonlinear material

- **Total displacement**
  - SOLS190: 27.96
  - SHELL181: 26.96
  - SOLID186: 22.51
  - SOLID187: 22.27
  - Fine Tets: 26

- **Stress**
  - SOLS190: 451
  - SHELL181: 453
  - SOLID186: 422
  - SOLID187: 402
  - Fine Tets: 453

- **Plastic strain**
  - SOLS190: 4.48
  - SHELL181: 4.63
  - SOLID186: 2.77
  - SOLID187: 2.41
  - Fine Tets: 4.4

- **Elapsed time**
  - SOLS190: 0
  - SHELL181: 0
  - SOLID186: 0
  - SOLID187: 0
  - Fine Tets: 0

© 2006 ANSYS, Inc.

ANSYS, Inc. Proprietary
3. Thin-wall structure simulation with nonlinear material

- SOLSH190
- SHELL181
- SOLID187(F)
- SOLID186
- SOLID187

Displacement

Stress

Plastic strain

$\rightarrow$
Possible ways for thin-wall simulation

1. Semi-automatic
   - Connection Technologies:
     - Bonding
     - Welding
     - Common nodes

2. Good surface mesh
   - SOLID-SHELL
   - SOLIDSHELL
   - SHELL
   - SHELLS
   - SOLIDS
   - SHELL MODELS
   - MPC

Note:
The connection between Solids and Shells can be achieved with MPC contact.
In order to solve large problems, 64-bit architecture is necessary, with that much more memory can be allocated.
5. Large models under Windows/XP 64-Bit

64 Bit Workbench

64 Bit Windows XP

Machine:

HP, CPU=AMD Opteron

RAM=16 GB, SWAP=32 GB

CPU=2
5. Large models under Windows/XP 64-Bit

Nonlinear static analysis

Elements: 486609  86 Iterations
Nodes:    770966  2 CPU: 9 h
DOFs:    3000000  Wall: 6 h
5. Large models under Windows/XP 64-Bit

Linear static analysis with bonded contact

Elements: 5.75 Mio.
Nodes: 8.66 Mio.
Contact: 36 k

DOFs: 26 Mio.
1 CPU: 3.6 h
Wall: 7.2 h
5. Large models under Windows/XP 64-Bit

Tets vs. Hex. and Shell

CAE process of acoustic analysis - BMW Otto Motor

Different FE models
- CAD
- Shells-Solids

- Hex.
- Tet.

Fig. 1 different ways of meshing of a four cylinder Otto Motor
5. Large models under Windows/XP 64-Bit

Tets vs. Hex. and Shell

Fig. 2 Correlation between FEA and tests for Shell-Solids model

Fig. 3 Correlation between FEA and tests for manually meshed Tet. model

Fig. 4 Correlation between FEA and tests for automatic Tet. Mesh model

Automatic Tet mesh

• Compare to Hex mesh, the Tet mesh greatly reduced the modeling time.
• and the accuracy of the results can also be improved with the Tet model.
5. Large models under Windows/XP 64-Bit

Nonlinear static analysis with contact

Elements: 3.05 Mio.
Nodes: 4.25 Mio.
Contact: 136 k
Iteration: 4

DOFs: 13 Mio.
2 CPU: 23h
Wall: 14h
5. Large models under Windows/XP 64-Bit

Nonlinear static analysis with contact local fine mesh

13 Iterations
2 CPU: 18 h
Wall: 11h

Elements: 1.4 Mio.
Nodes: 1.9 Mio.
Contact: 230 k
DOFs: 5.7 Mio.
5. Large models under Windows/XP 64-Bit

DOFs: 20 Mio.
5. Large models under Windows/XP 64-Bit
5. Large models under Windows/XP 64-Bit

64-bit Windows XP marks the new era of FEA, with the new memory opportunity, the engineers can finally say:

good-bye 2 GB limit!
6. Summary

The most efficient ways to solve linear and nonlinear thin-wall structure are:

1) Use DM to get mid-surface, use all-quad-mesh with SHELL181.
2) Use sweep mesh to get SOLSH190 model.

If the above methods can not be used, a fine SOLID187 model can also achieve the accurate result, but it’s a very expensive way.

Good knowledge and a good tool are the catalysts in solving the problems

- Good Knowledge + good Tool
- Problem
- Solution

- 20% - Mechanics expertise
- 20% - Engineer expertise
- 30% - FEA expertise
- 30% - Software expertise

© 2006 ANSYS, Inc.
7. Thank you!

Thanks

3d shell body
(continuous)

Shell
- shell model
  (discrete in $\xi$, continuous in $\eta$)
- finite element discretization
  2d shape functions
  (e.g. bilinear in $\xi, \eta$)
- directors
  6 DOF per node
  e.g. 4 nodes per element

Solid
- finite element discretization
  3d shape functions
  (e.g. tri-linear in $\xi, \eta, \zeta$)
- 3 DOF per node
  e.g. 8 nodes per element

Solid-Shell
- Special element formulation:
  e.g. assumed strain formulation
- 3 DOF per node
  e.g. 8 nodes per element