



ANSYS_LS-DYNA跌落仿真分析

—跌落模块、模型简化和单元选择

吴华春

内容

一、简介



二、ANSYS Drop Test Wizard

三、模型简化规则

四、单元类型及选择

五、计算结果评估

1.简介

1.1 跌落仿真应用说明

跌落仿真分析随着CAE技术渗透各行各业，其可以准确模拟出各行各业的产品结构的受力和变形情况，依此评估产品的设计缺陷，优化产品的结构及质量，缩短产品的生产周期，极大的节约产品成本。



1.2. 跌落仿真应用分类

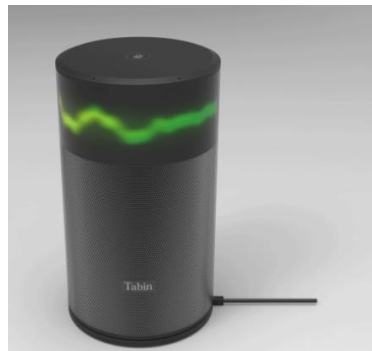
按照有无包装进行分类

1.2.1 无包装产品跌落

主要在日常生活中，经常使用的产品以及产品制造装配中的零部件以及军工产品。其细分行业如下：

1.2.1.1 便携式电子产品

其包含手机、笔记本电脑、智能音箱、遥控器等。



1.2. 跌落仿真应用分类

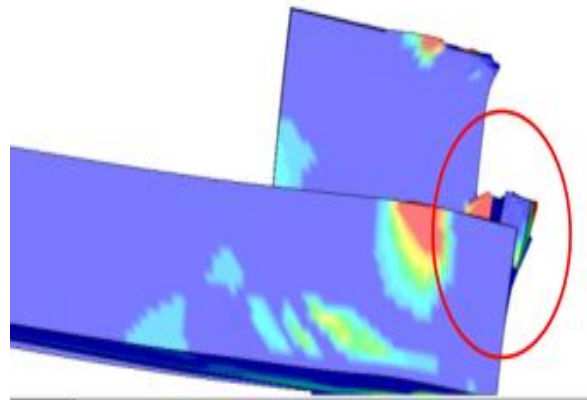
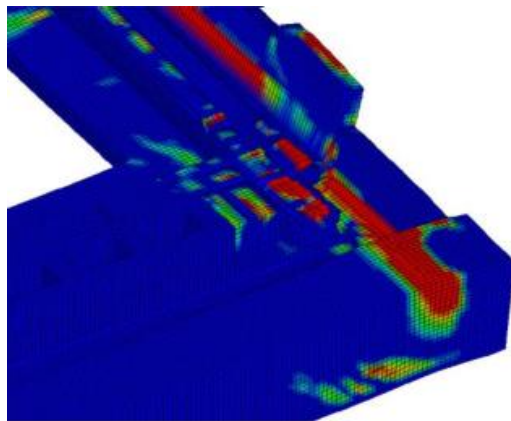
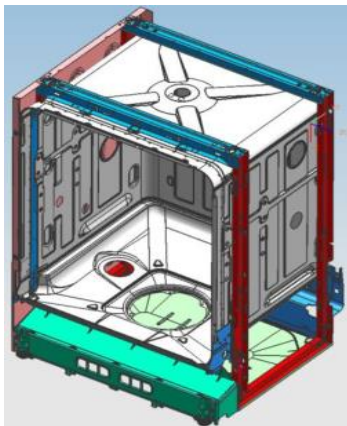
按照有无包装进行分类

1.2.2 包装产品跌落

主要在产品运输过程。

1.2.2.1 家电产品

冰箱、洗衣机、抽油烟机、空调、电饭煲、微波炉、洗碗机等家电产品。



1.2. 跌落仿真应用分类

1.2.2.2 包含液体的包装盒及瓶



内容

一、简介

二、ANSYS Drop Test Wizard



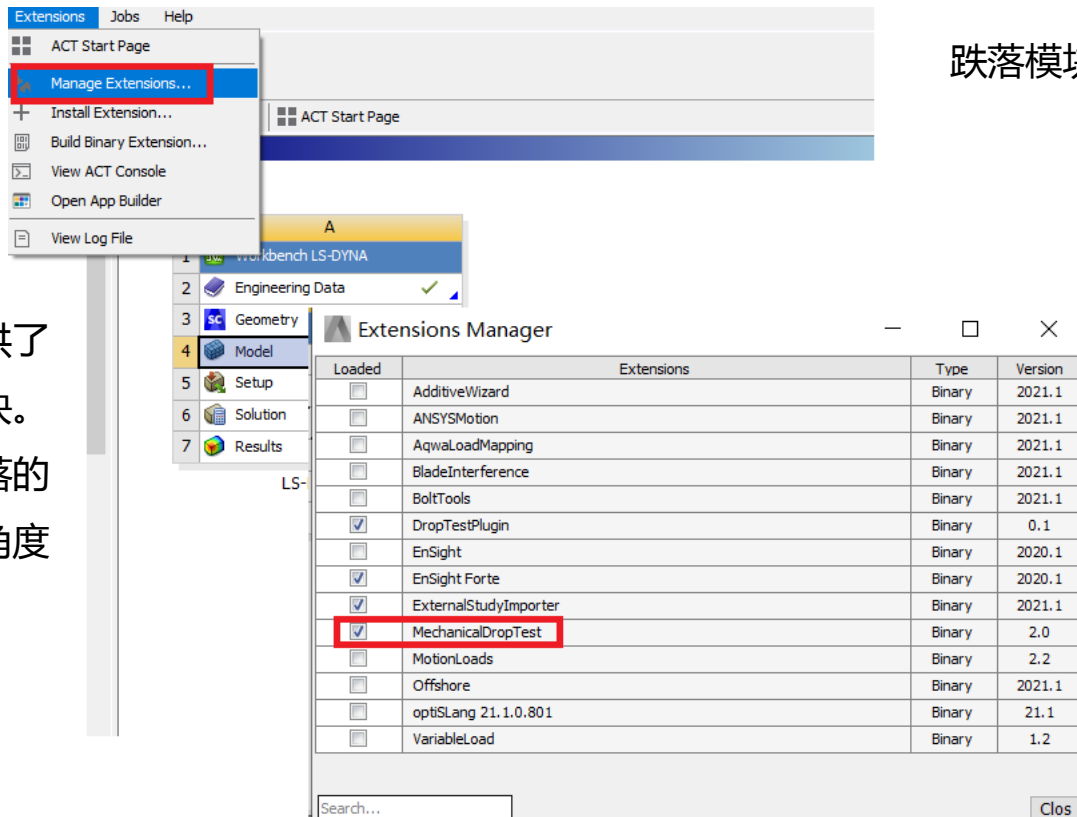
三、模型简化规则

四、单元类型及选择

五、计算结果评估

2.1. 跌落模块调用

- ◆ ANSYS LS-DYNA提供了跌落的全自动设置模块。
- ◆ 用户只需提供需要跌落的几何以及跌落高度、角度参数。



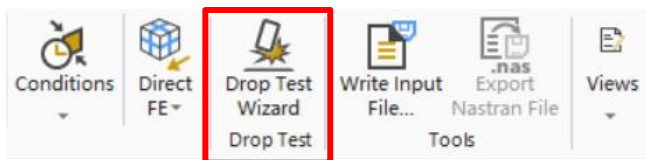
Extensions Manager

| Loaded | Extensions | Type | Version |
|-------------------------------------|-----------------------|--------|---------|
| <input type="checkbox"/> | AdditiveWizard | Binary | 2021.1 |
| <input type="checkbox"/> | ANSYSMotion | Binary | 2021.1 |
| <input type="checkbox"/> | AqwaLoadMapping | Binary | 2021.1 |
| <input type="checkbox"/> | BladeInterference | Binary | 2021.1 |
| <input type="checkbox"/> | BoltTools | Binary | 2021.1 |
| <input checked="" type="checkbox"/> | DropTestPlugin | Binary | 0.1 |
| <input type="checkbox"/> | EnSight | Binary | 2020.1 |
| <input checked="" type="checkbox"/> | EnSight Forte | Binary | 2020.1 |
| <input checked="" type="checkbox"/> | ExternalStudyImporter | Binary | 2021.1 |
| <input checked="" type="checkbox"/> | MechanicalDropTest | Binary | 2.0 |
| <input type="checkbox"/> | MotionLoads | Binary | 2.2 |
| <input type="checkbox"/> | Offshore | Binary | 2021.1 |
| <input type="checkbox"/> | optiSLang 21.1.0.801 | Binary | 21.1 |
| <input type="checkbox"/> | VariableLoad | Binary | 1.2 |

跌落模块的调用。

2.1. 跌落模块调用

跌落模块的调用。



Wizard

Drop Test

ANSYS ACT

| | | |
|---------------------|-------------|-----|
| Target Rotation (X) | 0 | deg |
| Drop Rotation (X) | -25 | deg |
| Drop Rotation (Y) | 0 | deg |
| Drop Rotation (Z) | 0 | deg |
| Define By | Drop Height | |
| Drop Height | 1 | m |

Help

Target Rotation (X) orients the automatically generated Target geometry about the global X axis. The Target geometry is created on entering a non-zero value into this input field.

The orientation of the geometry to be dropped is specified using the Drop Rotation (X), Drop Rotation (Y), and Drop Rotation (Z) input fields. Drop rotations are about the center of mass of the dropped geometry and are applied instantaneously to allow immediate visual inspection.

Impact conditions can be defined by either Drop Height or Impact Velocity.

Drop Height reflects the height from which the object is dropped where it undergoes gravitational acceleration.

Exit Wizard Back Next

- ❑ 创建刚性地面，表示物体下落的平面
- ❑ 创建模型和下落平面的方向
- ❑ 设置跌落高度，自动计算模型的冲击速度
- ❑ 加载标准地球重力
- ❑ 完全限制刚性地面
- ❑ 自动定义刚性地面和模型之间的接触
- ❑ 自动划分模型网格
- ❑ 设置分析结束时间
- ❑ 自动设置部分历史进程曲线
- ❑ 自动设置动画输出间隔
- ❑ 等等，具体自动设置内容见后面PPT。

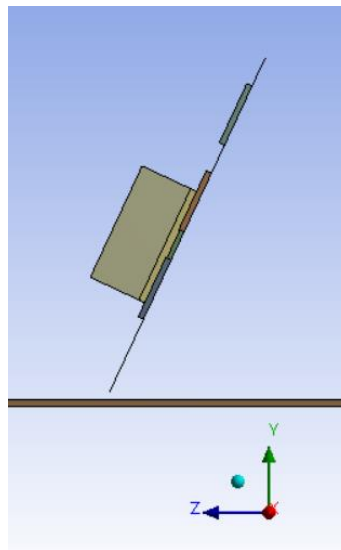
2.2. 跌落模块简单设置

单击模型树中的LS-DYNA分支，并从工具栏的环境界面中单击拖放跌落向导

- 将创建一个垂直于全局Y轴的刚性地面，因此保持目标旋转(X) = 0
- 电路板将以25°的角度冲击，所以跌落旋转(X) = -25°

注意，图形更新允许预览下拉旋转

- 设置跌落高度= 1米
- 然后，在“跌落向导”中查看信息
- 单击Next。



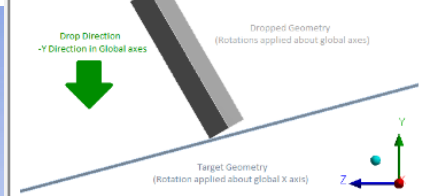
Wizard

Drop Test

ANSYS ACT

| | | |
|---------------------|-------------|-----|
| Target Rotation (X) | 0 | deg |
| Drop Rotation (X) | -25 | deg |
| Drop Rotation (Y) | 0 | deg |
| Drop Rotation (Z) | 0 | deg |
| Define By | Drop Height | |
| Drop Height | 1 | m |

Help



Target Rotation (X) orients the automatically generated Target geometry about the global X axis. The Target geometry is created on entering a non-zero value into this input field.

The orientation of the geometry to be dropped is specified using the Drop Rotation (X), Drop Rotation (Y), and Drop Rotation (Z) input fields. Drop rotations are about the center of mass of the dropped geometry and are applied instantaneously to allow immediate visual inspection.

Impact conditions can be defined by either Drop Height or Impact Velocity.

Drop Height reflects the height from which the object is dropped where it undergoes gravitational acceleration.

Exit Wizard Back Next

2.2. 跌落模块简单设置

- 选择摩擦行为
- 摩擦系数= 0.1
- 然后，检查向导中的其余信息
- 单击Finish

Wizard

Drop test

Drop Test

ANSYS ACT

Frictional Behavior: Frictional

Friction Coefficient: 0.1

Dynamic Coefficient: 0

Help

Drop Direction
-Y Direction in Global axes

Dropped Geometry
(Rotations applied about global axes)

Target Geometry
(Rotation applied about global X axis)

Frictional Behavior is used to create a global body interaction defining contact between the dropped geometry and the Target geometry.

After clicking Finish:

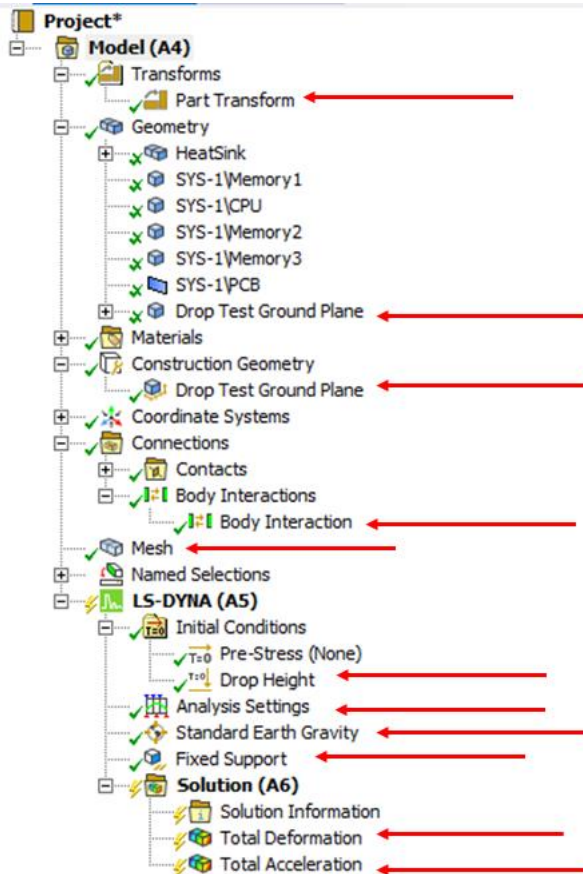
- The global body interaction is added to the analysis.
- A Standard Earth Gravity boundary condition scoped to all bodies is added to the analysis.
- The mesh is generated.
- A check is performed to ensure that there are no initial penetrations between the dropped geometry and the Target geometry.
- Three result objects are added to the the analysis.

Exit Wizard Back Finish

2.3. 跌落模块自动设置内容

查看跌落向导添加的参数:

- 添加了一个零件移动，以确定下落的几何形状
- 创建地面几何，并添加刚性下降测试地面平面几何分支
- 定义了一个模型和地面的摩擦
- 根据前面指定的尺寸定义生成一个Mesh
- 添加了一个下落高度初始条件与计算的冲击速度
- 设置分析结束时间
- 增加了一个地面的约束
- 增加了地球重力(-Y)
- 增加变形动画结果，加速度动画结果



内容

一、简介

二、ANSYS Drop Test Wizard

三、模型简化规则



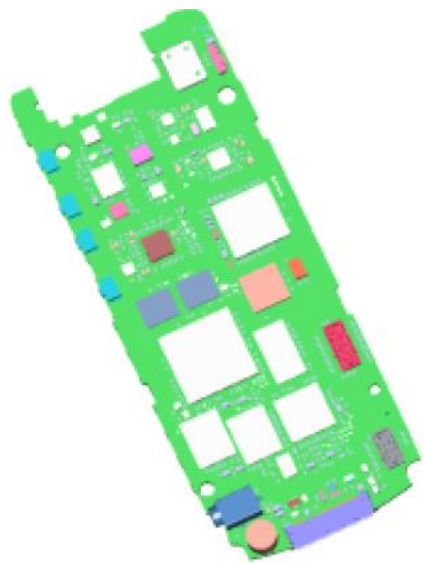
四、单元类型及选择

五、计算结果评估

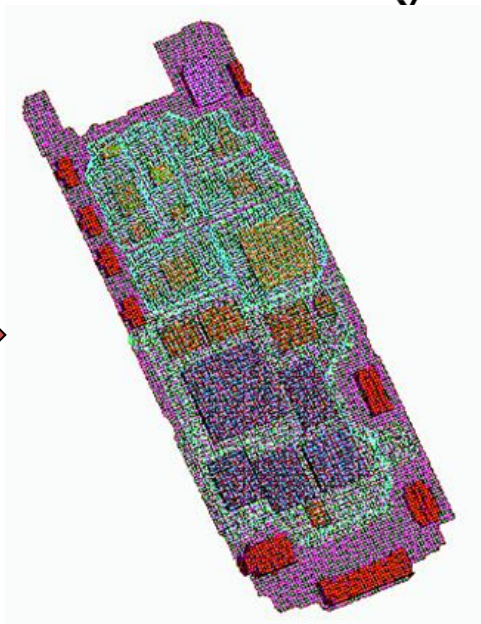
3. 模型简化规则

下面以电子产品为例进行说明。

一部手机由许多部件组成，如果将所有部件建模，将需要大量的单元。此外，一些单元会非常小，由于一些微小的组成部分和细节特征。如右图所示，为了减少CPU时间，必须进行简化。



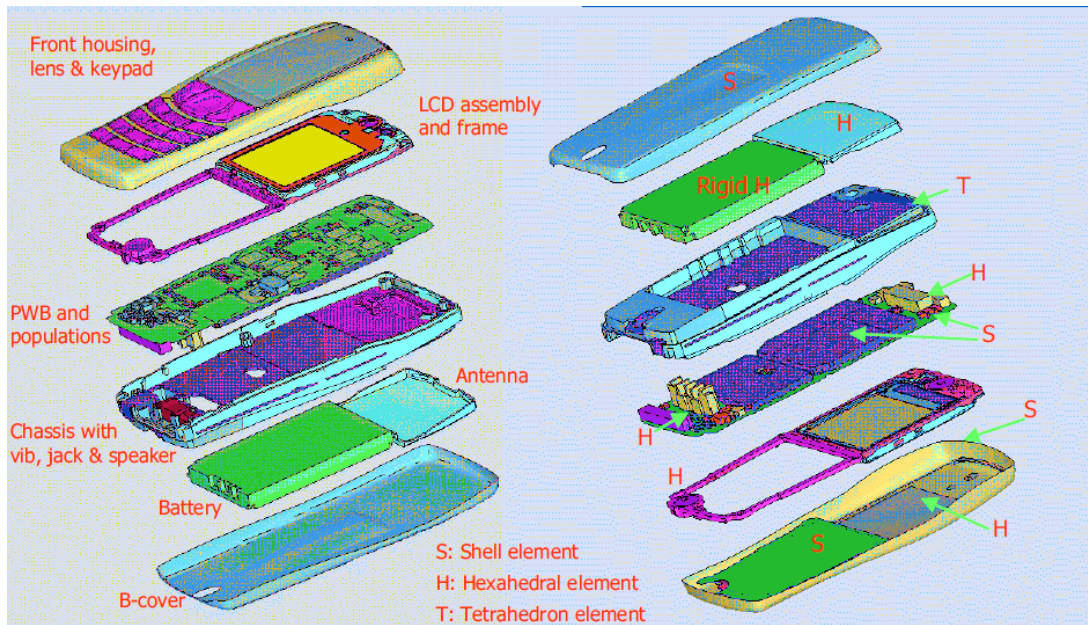
设计模型



FEA模型

3. 模型简化规则

下面以电子产品为例进行说明。

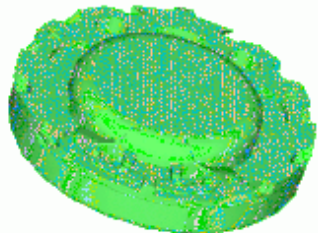


3. 模型简化规则

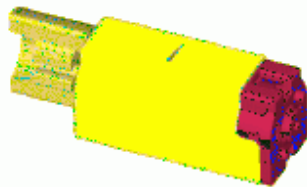
下面以电子产品为例进行说明。

- ◆ 一些细小的部件，很薄的薄膜，磁带都可以忽略
- ◆ 简化非重要部件的建模。可以用质量点代替，记住：质量点需要均布。

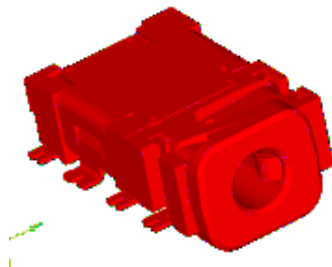
CAD模型



1119个面

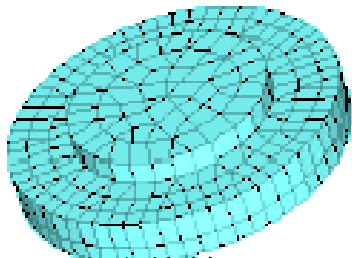


559个面

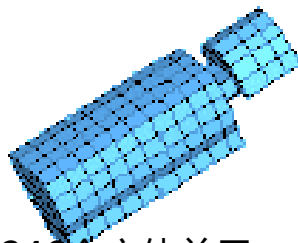


1643个面

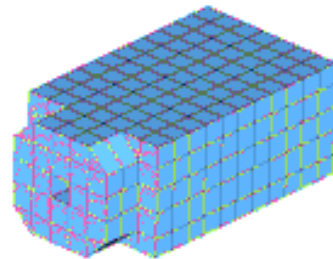
FEA模型



531个实体单元



246个实体单元

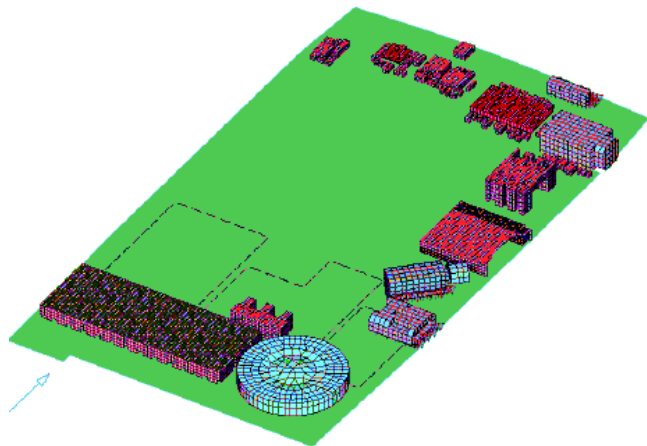


422个实体单元

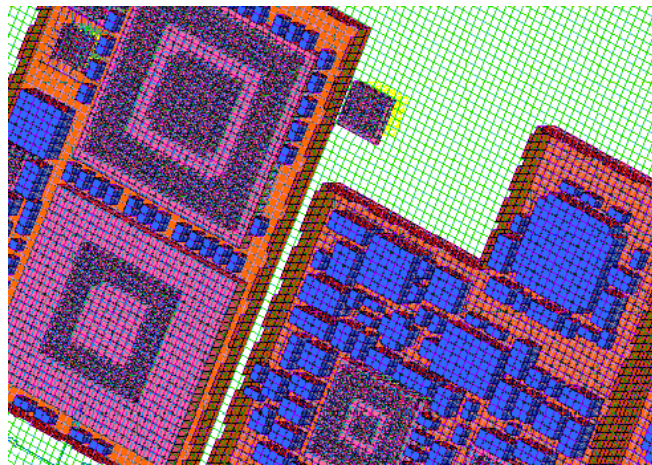
3. 模型简化规则

下面以电子产品为例进行说明。

- ◆ 小的功能被忽略，但有些部分不能被删除。这些是对刚度，质量，惯性，接触等有影响的。
 - 一个背壳由2000个表面组成(例如，有1903个表面在手机)。
 - 一个完整的PWB板是定义了100K的表面。
- ◆ 连接部件——螺丝、削波器.....——可以在ANSYS LS-DYNA中通过刚性梁建模。bond接触或节点刚体也可用于简化连接。



简化后的PWB板

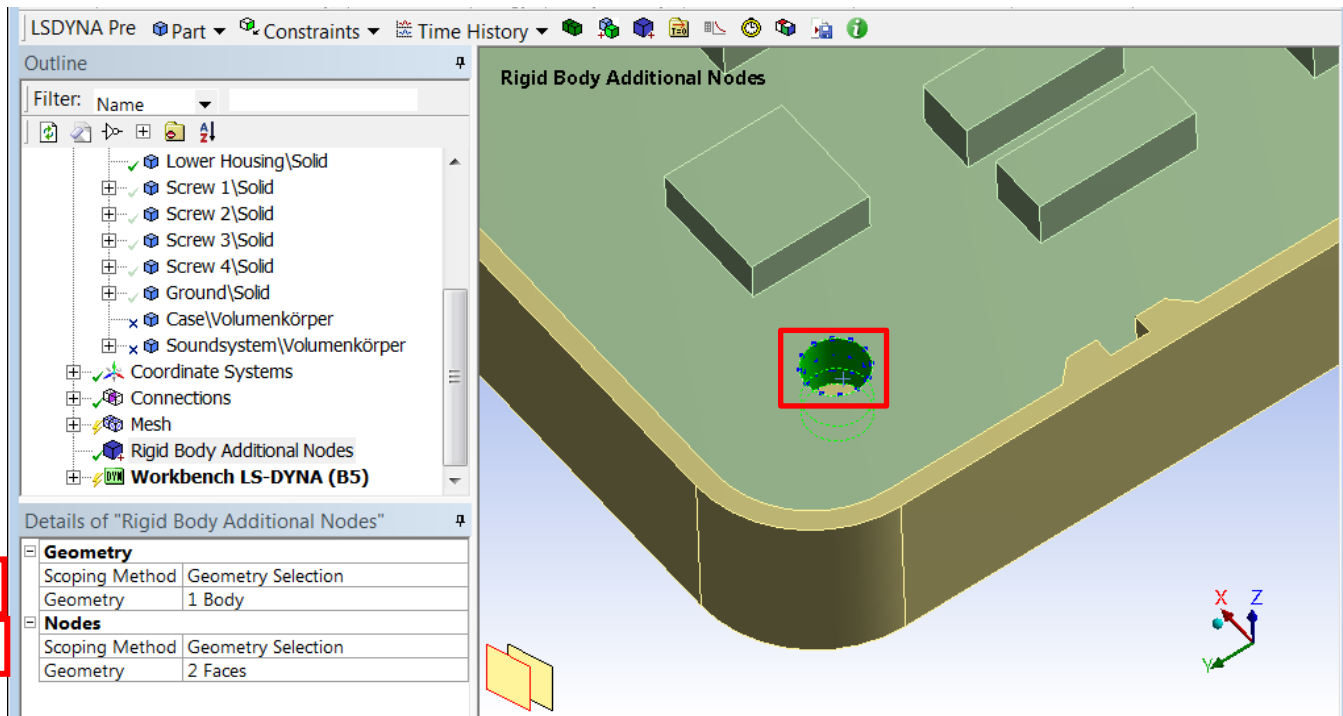


芯片组封装，球栅阵列和shields的详细建模

3. 模型简化规则

下面以电子产品为例进行说明。

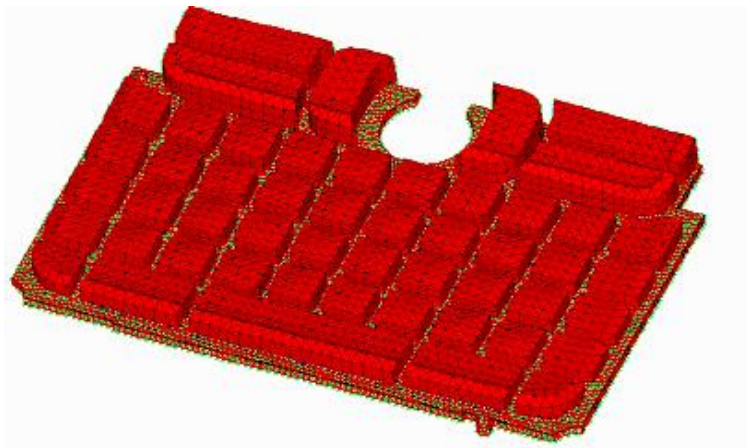
◆ 刚性连接



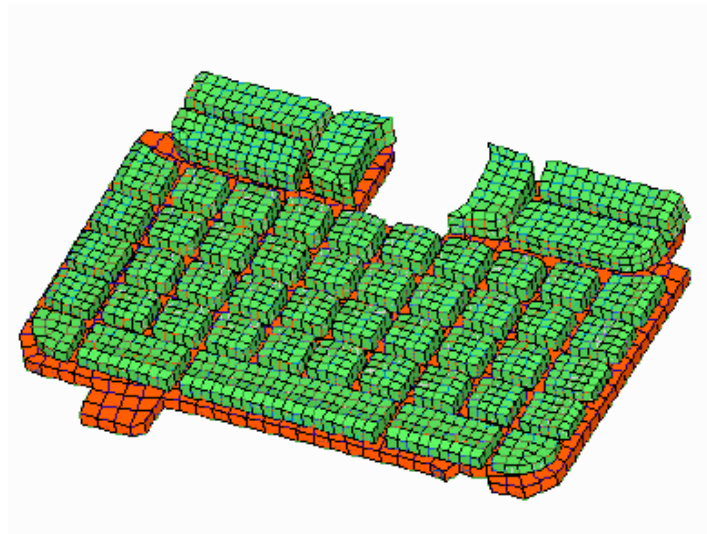
注：此处由4个螺栓，每个螺栓依次建立Rigid Body Additional Nodes

3. 模型简化规则

下面以电子产品为例进行说明。



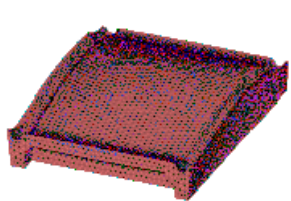
103,703个10节点四面体单元



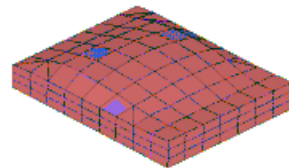
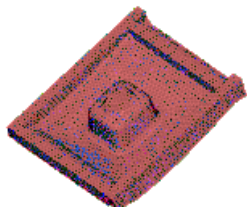
3668个六面体单元

3. 模型简化规则

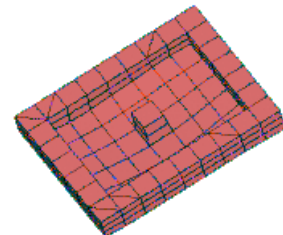
下面以电子产品为例进行说明。



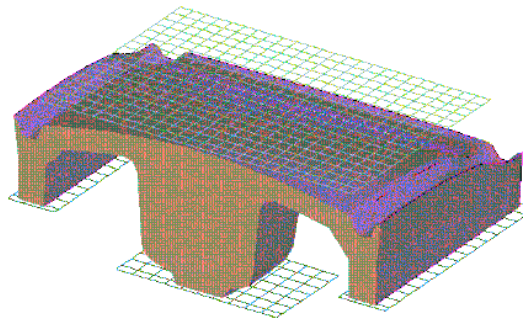
31234 四面体单元



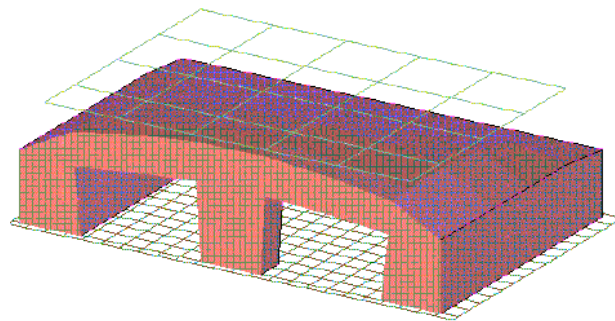
139 六面体



20 ms 压缩试验



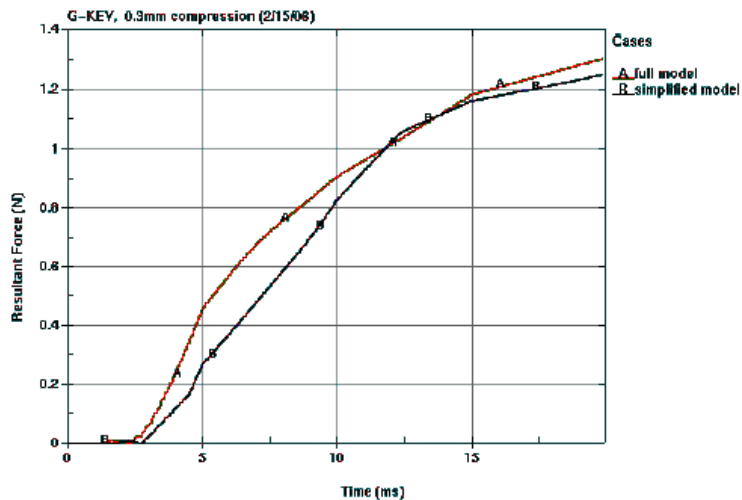
Full model: 5:17'12"



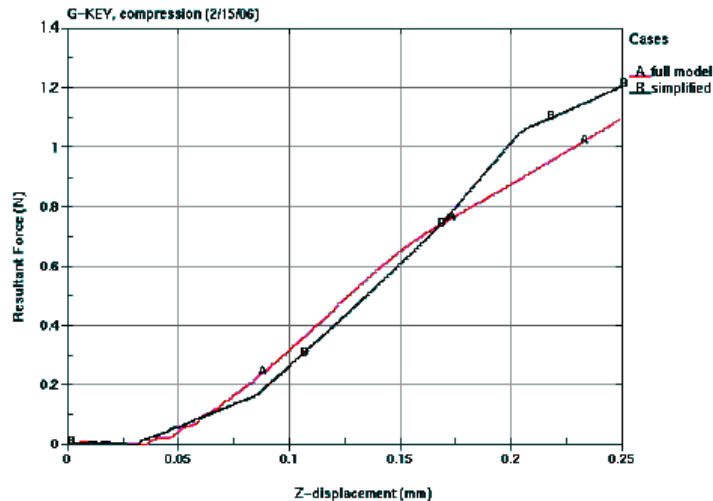
Simplified model: 15"

3. 模型简化规则

下面以电子产品为例进行说明。



接触力对比



刚度比较

结论：

刚度差小于10%。

对于硅树脂这样的软材料，六面体单元在跌落模拟中更加稳定。

3.模型简化规则

总结：

- 1) 记住有限元分析是搜索微分方程组的解，而不是动画显示。最重要的一点是在建模过程中保持模拟对象的基本力学特性，而不是几何细节。
- 2) 简化的程度取决于模拟的目的。简化必须在一定的水平上，使模型能够涵盖跌落过程中所有的力学行为的细节。

内容

一、简介

二、ANSYS Drop Test Wizard

三、模型简化规则

四、单元类型及选择



五、计算结果评估

在ANSYS LS-DYNA软件中可用单元模型，包括ALE单元、Beam单元、SEATBELT（安全带）单元、Shell单元、Solid单元等近120多种等单元类型，这对于刚熟悉LS-DYNA的用户来说，如何进行单元选择仍然是个不小的难题。

| LS-DYNA(单元积分公式种类) | |
|-------------------------------|-------|
| *SECTION_ALE1D (3) | ALE |
| *SECTION_ALE2D (2) | ALE |
| *SECTION_BEAM (14) | BEAM |
| *SECTION_BEAM_AISC (7) | BEAM |
| *SECTION_DISCRETE (与材料结合) | BEAM |
| *SECTION_POINT_SOURCE | |
| *SECTION_POINT_SOURCE_MIXTURE | |
| *SECTION_SEATBELT (1) | SHELL |
| *SECTION_SHELL (47) | SHELL |
| *SECTION_SOLID (42) | SOLID |
| *SECTION_SPH | SPH |
| *SECTION_TSHELL (6) | SHELL |

在ANSYS LS-DYNA跌落模型中，通常只使用几种类型的单元。

在跌落模型中，就像整个手机模型一样，必须使用各种元素类型。单元类型的选择取决于，例如：

- 零件几何形状和变形模式。
- 单元性能和效率。ANSYS LS-DYNA有一个大的元素库和每个元素都有它的独特的特点。

对于不同的单元组，有不同的类型可以与跌落测试模拟相关：

3D 单元:

Hexahedral: Type 1, 2, 3

Pentahedron: Type 15

Four noded tetrahedron: Type 4 & 10

Ten noded tetrahedron: Type 16 & 17

2D 单元:

Thin shells: Type 1-11, 16-17 and 23-24.

Thick shells: Type 1, 2 and 3 (3D).

1D 单元:

Beams: Type 1-6, 9 and 11-12.

Springs

在ANSYS LS-DYNA跌落模型中，通常只使用几种类型的单元。

可以看到有许多不同的单元类型。然而，通常只使用其中的少数几种。跌落试验模拟和其它模型也是如此。

本文中使用的单元如下:

3D 单元:

Hexahedral: Type 1, 2, 3

Pentahedron: Type 15, 通过在*CONTROL_SOLID上使用ESORT自动分配。

Four noded tetrahedron: Type 4 & 10

Ten noded tetrahedron: Type 16

2D 单元:

Thin shells: Type 2.

Thick shells: Type 1

1D 单元:

Beams: Type 1.

Springs

| *SECTION_SOLID_(TITLE) (8) | | |
|----------------------------|--------|-----|
| TITLE | | |
| <input type="text"/> | | |
| SECID | ELFORM | AET |
| 1 | 10 | 0 |

EQ.0: 1 point corotational for *MAT_MODIFIED_HONEY-COMB (see [Remark 3](#))

EQ.1: constant stress solid element: default element type. By specifying hourglass type 10 with this element, a Cosserat Point Element is invoked, see *CONTROL_HOURLASS

EQ.2: 8 point hexahedron (see [Remark 4](#))

EQ.3: Fully integrated quadratic 8 node element with nodal rotations

EQ.4: S/R quadratic tetrahedron element with nodal rotations

EQ.6: 1 point Eulerian

EQ.7: 1 point Eulerian ambient

EQ.8: Acoustic

EQ.9: 1 point corotational for *MAT_MODIFIED_HONEY-COMB (see [Remark 3](#))

EQ.10: 1 point tetrahedron (see [Remark 1](#))

EQ.11: 1 point ALE multi-material element

EQ.12: 1 point integration with single material and void

EQ.13: 1 point nodal pressure tetrahedron (see [Remark 14](#))

EQ.14: 8 point acoustic

EQ.15: 2 point pentahedron element (see [Remark 1](#))

EQ.16: 4 or 5 point 10-noded tetrahedron (see [Remark 13](#)). By specifying hourglass type 10 with this element, a Cosserat Point Element is invoked; see *CONTROL_HOURLASS.

EQ.17: 10-noded composite tetrahedron (see [Remark 13](#))

EQ.18: 8 point enhanced strain solid element for linear statics only (see [Remark 4](#))

- LS-DYNA常用的六面体单元类型 (Element formulation)

常用的类型有-2、 -1 , 1 , 2 , 最常用的为1.

- LS-DYNA常用的四面体单元类型 (Element formulation)

| *SECTION_SOLID_(TITLE) (8) | | |
|------------------------------|--------|-----|
| TITLE | | |
| <input type="text"/> | | |
| SECID | ELFORM | AET |
| 1 | 10 | 0 |

- EQ.0: 1 point corotational for *MAT_MODIFIED_HONEY-COMB (see [Remark 3](#))
- EQ.1: constant stress solid element: default element type. By specifying hourglass type 10 with this element, a Cosserat Point Element is invoked, see *CONTROL_HOURLASS
- EQ.2: 8 point hexahedron (see [Remark 4](#))
- EQ.3: Fully integrated quadratic 8 node element with nodal rotations
- EQ.4: S/R quadratic tetrahedron element with nodal rotations

常用的类型有4、10、13、16、17，最常用的为4，10.

- EQ.6: 1 point Eulerian
- EQ.7: 1 point Eulerian ambient
- EQ.8: Acoustic
- EQ.9: 1 point corotational for *MAT_MODIFIED_HONEY-COMB (see [Remark 3](#))
- EQ.10: 1 point tetrahedron (see [Remark 1](#))
- EQ.11: 1 point ALE multi-material element
- EQ.12: 1 point integration with single material and void
- EQ.13: 1 point nodal pressure tetrahedron (see [Remark 14](#))
- EQ.14: 8 point acoustic
- EQ.15: 2 point pentahedron element (see [Remark 1](#))
- EQ.16: 4 or 5 point 10-noded tetrahedron (see [Remark 13](#)). By specifying hourglass type 10 with this element, a Cosserat Point Element is invoked; see *CONTROL_HOURLASS.
- EQ.17: 10-noded composite tetrahedron (see [Remark 13](#))
- EQ.18: 8 point enhanced strain solid element for linear statics only (see [Remark 4](#))

• LS-DYNA常用的shell单元类型 (Element formulation)

| *SECTION_SHELL_(TITLE) (6) | | | | | | | | | |
|------------------------------|-----------|-----------|----------------------------|-----------|------|---------|-------|--------|--|
| TITLE | | | | | | | | | |
| SHELL-0.07mm | | | | | | | | | |
| SECID | ELFORM | SHR | Enter data into text field | | ROPT | QR/TRID | ICOMP | SETYP | |
| 1 | 22 | 0 | 1.0000000 | 2 | 1 | 0 | 0 | 1 | |
| 2 | T1 | T2 | T3 | T4 | HLQC | MAREA | IDOF | EDGSET | |
| | 0.0700000 | 0.0700000 | 0.0700000 | 0.0700000 | 0.0 | 0.0 | 0.0 | 0 | |

EQ.1: Hughes-Liu

EQ.2: Belytschko-Tsay

EQ.3: BCIZ triangular shell

EQ.4: C0 triangular shell

EQ.5: Belytschko-Tsay membrane

EQ.6: Selectively reduced Hughes-Liu

EQ.7: Selectively reduced, co-rotational Hughes-Liu

EQ.8: Belytschko-Leviathan shell

EQ.9: Fully integrated Belytschko-Tsay membrane

EQ.10: Belytschko-Wong-Chiang

常用的类型有2,16, 最常用的为2

EQ.11: Fast (co-rotational) Hughes-Liu

EQ.12: Plane stress (xy -plane)

EQ.13: Plane strain (xy -plane)

EQ.14: Axisymmetric solid (xy -plane, y -axis of symmetry) - area weighted (see Remark 10)

EQ.15: Axisymmetric solid (xy -plane, y -axis of symmetry) - volume weighted

EQ.16: Fully integrated shell element (very fast)

EQ.-16: Fully integrated shell element modified for higher accuracy. See Remark 13.

EQ.17: Fully integrated DKT, triangular shell element. See Remark 9.

EQ.18: Fully integrated linear DK quadrilateral/triangular shell

EQ.20: Fully integrated linear assumed strain C0 shell. See Remark 2.

EQ.21: Fully integrated linear assumed strain C0 shell (5 DOF)

EQ.22: Linear shear panel element. 3 DOF per node. See Remark 3.

EQ.23: 8-node quadratic quadrilateral shell (see IRQUAD in *CONTROL_SHELL)

EQ.24: 6-node quadratic triangular shell

EQ.25: Belytschko-Tsay shell with thickness stretch

EQ.26: Fully integrated shell with thickness stretch

EQ.27: C0 triangular shell with thickness stretch

EQ.29: Cohesive shell element for edge-to-edge connection of shells. See Remark 12.

EQ.-29: Cohesive shell element for edge-to-edge connection of shells (more suitable for pure shear). See Remark 12.

EQ.41: Mesh-free (EFG) shell local approach (more suitable for crashworthiness analysis).

EQ.42: Mesh-free (EFG) shell global approach (more suitable for metal forming analysis).

EQ.43: Mesh-free (EFG) plane strain formulation (xy -plane)

EQ.44: Mesh-free (EFG) axisymmetric solid formulation (xy -plane, y -axis of symmetry)

EQ.46: Cohesive element for two-dimensional plane strain, plane stress, and area-weighted axisymmetric problems (use with type 14 shells). See Remark 15.

EQ.47: Cohesive element for two-dimensional volume-weighted axisymmetric problems (use with type 15 shells). See Remark 15.

EQ.52: Plane strain (xy -plane) XFEM, base element type 13

EQ.54: Shell XFEM, base element type defined by BASELM (default 2)

EQ.55: 8-node singular plane strain (xy -plane) finite element. See Remark 11.

EQ.98: Interpolation shell

EQ.99: Simplified linear element for time-domain vibration studies. See Remark 4.

EQ.101: User defined shell

EQ.102: User defined shell

EQ.103: User defined shell

EQ.104: User defined shell

EQ.105: User defined shell

EQ.201: Isogeometric shells with NURBS. See *ELEMENT_SHELL_NURBS_PATCH.

GE.1000: Generalized shell element formulation (user defined). See *DEFINE_ELEMENT_GENERALIZED_SHELL.

- LS-DYNA常用的T-shell单元类型 (Element formulation)

*SECTION_TSHELL_(TITLE) (0)

TITLE

SECID ELFORM SHRE NIP PROPT QR ICOMP TSHEAR

| | | | | | | | |
|---|---|-----|-----|-----|---|---|---|
| 1 | 1 | 1.0 | 2.0 | 1.0 | 0 | 0 | 0 |
|---|---|-----|-----|-----|---|---|---|

EQ.1: one point reduced integration (default),

EQ.2: selective reduced 2×2 in plane integration.

EQ.3: assumed strain 2×2 in plane integration, see remark below.

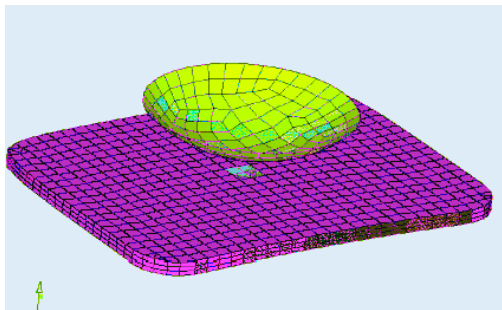
EQ.5: assumed strain reduced integration with brick materials

EQ.6: assumed strain reduced integration with shell materials

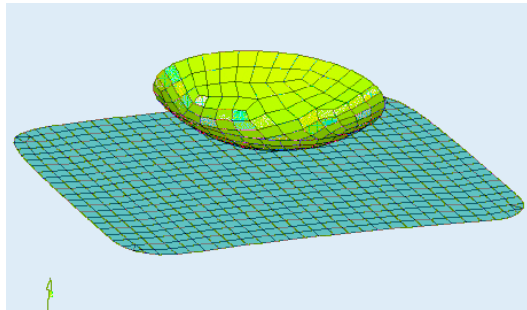
EQ.7: assumed strain 2×2 in plane integration.

常用的类型有 1.

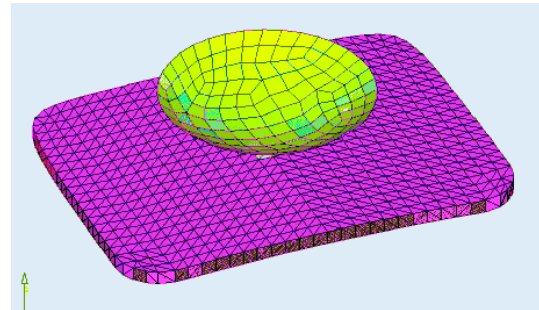
• 单元对比测试1---落球试验



3-layer hex elements



shell elements



1-layer tetra elements

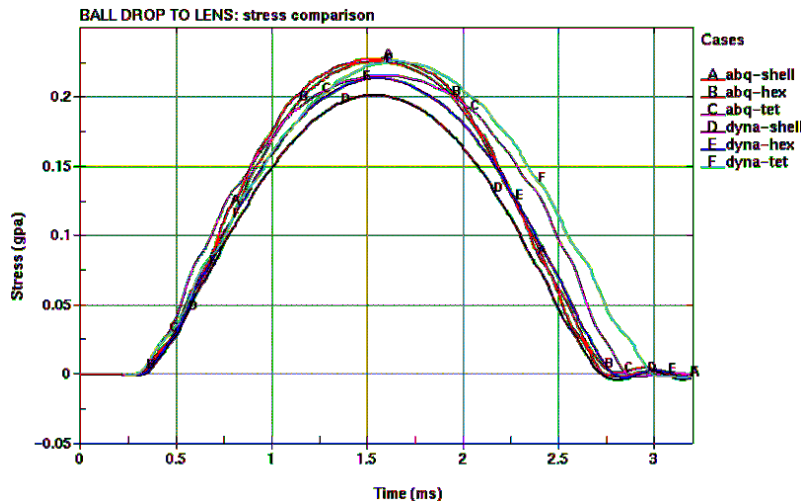
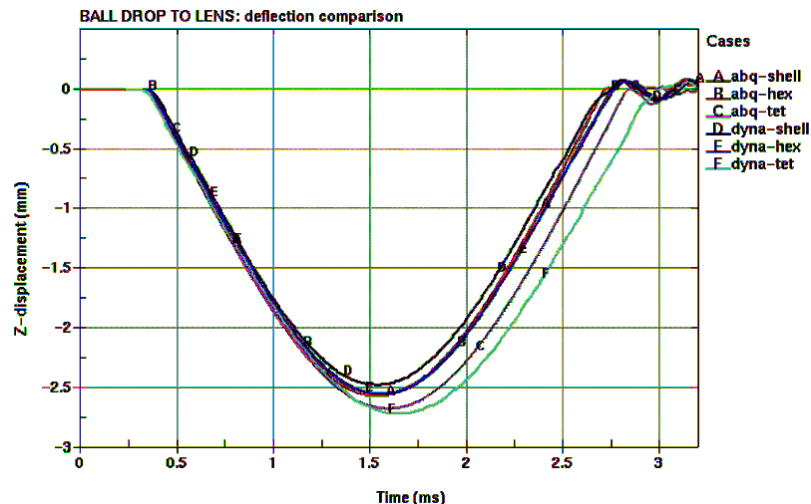
基准条件：

- ◆ 球落在—块面板上。弹性材料模型，挠度大。
- ◆ 通过厚度使用9个高斯点。这使得高斯积分具有较高的精度，对于非线性模型和塑性模型尤为重要。使用大量高斯积分点会带来CPU损失，在这个模型中，与使用默认高斯积分点的模型相比，CPU损失约为10%。壳的应力输出不是在表面，而是在高斯点，这可以用INTGRD在*CONTROL_SHELL改变。
- ◆ 应力数据采用截止频率为4khz的LS-PrePost®BW滤波器进行滤波。

单元对比测试1---落球试验

| Element# | Deflection(mm) | XX-stress(GPa) | CPU(s) |
|----------------|----------------------|-----------------------|--------------------|
| Shell 2 | 2.472 / -3.0% | 0.207* / -3.7% | 29 / 1.0 |
| Solid 1 | 2.548 / 0.0% | 0.215 / 0.0% | 95 / 3.3 |
| Tetrahedron 16 | 2.712 / +6.4% | 0.227 / +5.6% | 1517 / 52.3 |

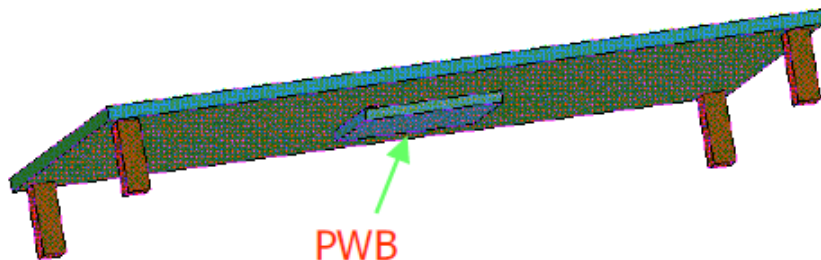
注意:应力相对于实体类型1, 而CPU时间相对于shell 2。



• 单元对比测试2---PWB板跌落测试

Compare element types in PWB (Printed Wiring Board)

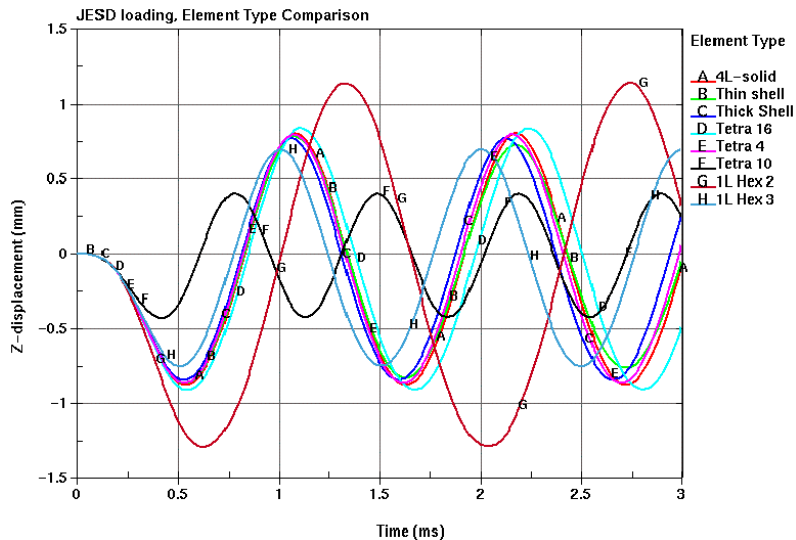
- A. 4-layer Hexahedral (type 1)
- B. Thin shell (type 2)
- C. Thick shell (type 1)
- D. Tetrahedron 16 (10-node)
- E. Tetrahedron 4 (4-node)
- F. Tetrahedron 10 (4-node)
- G. 1-layer full-integral-2 hexahedral
- H. 1-layer full-integral-3 hexahedral



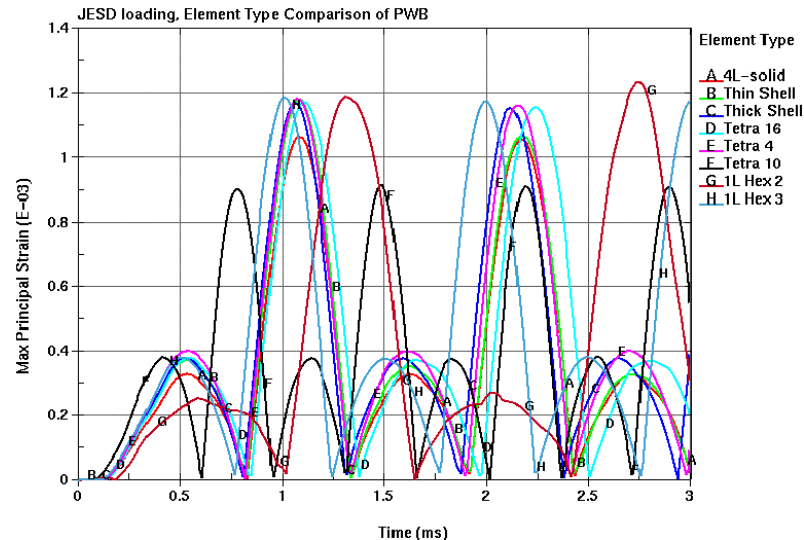
模型描述：

- ◆ 四面体模型包含15,004个四面体单元，壳/六面体模型包含10,965个壳/六面体单元，还有625个壳单元(芯片组包CSP)和288beam(焊接点)。
- ◆ 3 ms仿真，采用1500g/0.5ms半正弦基加速度，应用于* BOUNDARY_PRESCRIBED_MOTION_SET选项

• 单元对比测试2---PWB板跌落测试Max principal strain at PWB center



Deflection at PWB center



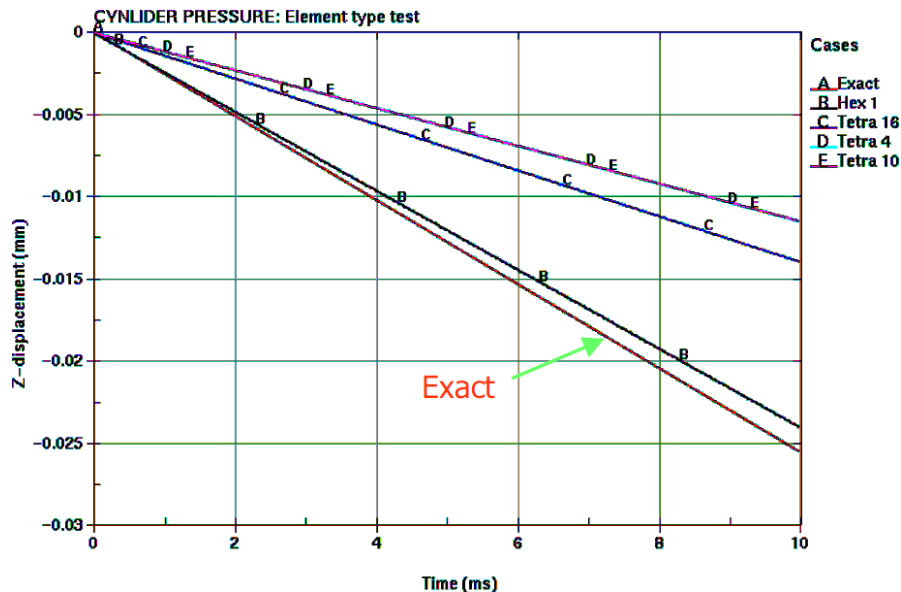
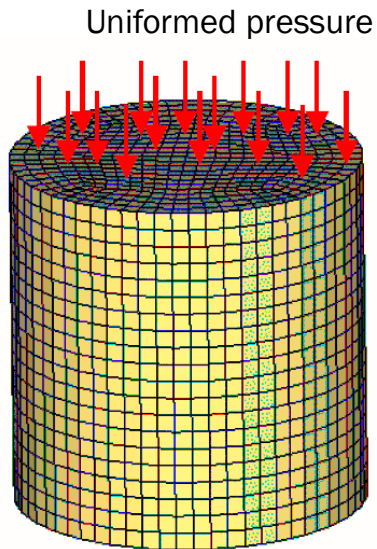
Compare element types in PWB

• 单元对比测试2---PWB板跌落测试Max principal strain at PWB center

| Element type | 4-L Hex(A) | Thin shell(B) | Thick shell(C) | Tetra 16(D) | Tetra 4(E) | Tetra 10(F) | 1L-Hex 2(G) | 1L-Hex3(H) |
|-----------------------|------------|---------------|----------------|-------------|------------|-------------|-------------|------------|
| Max deflection (-) mm | 0.876 | 0.865 | 0.841 | 0.911 | 0.861 | 0.426 | 1.288 | 0.7503 |
| Max deflection (+) mm | 0.807 | 0.787 | 0.775 | 0.840 | 0.800 | 0.408 | 1.146 | 0.6974 |
| Max strain (x0.001) | 1.065 | 1.180 | 1.170 | 1.174 | 1.183 | 0.916 | 1.236 | 1.186 |
| CPU time (min) | 35 | 11 | 25 | 84 | 35 | 9 | 22 | 61 |

| Element type | 4-L Hex(A) | Thin shell(B) | Thick shell(C) | Tetra 16(D) | Tetra 4(E) | Tetra 10(F) | 1L-Hex 2(G) | 1L-Hex3(H) |
|--------------|------------|---------------|----------------|--------------|------------|-------------|-------------|-------------|
| Stiffness | ✓ | ✓ | ✓ | a bit stiff | ✓ | much stiff | much soft | a bit stiff |
| Strain | low | ✓ | ✓ | ✓ | ✓ | much low | ✓ | ✓ |
| Phase | ✓ | ✓ | ✓ | a bit behind | ✓ | much ahead | much behind | a bit ahead |
| Damping | ✓ | effect | ✓ | ✓ | ✓ | ✓ | ✓ | ✓ |
| Efficiency | 35 | 11 | 25 | 84 | 35 | 9 | 22 | 61 |

• 单元对比测试3---压缩测试



从简单的压缩测试:

- ◆ 六面体单元1具有最好的性能，非常接近精确解。
- ◆ 四面体16给出了一个较硬的响应。在本试验案例中，四面体4和四面体10比四面体16更坚硬。

四、单元类型及选择-----总结

- ◆ 单元选择直接关系到精度和CPU成本。
- ◆ 如果弯曲变形的薄壁零件占优势，则首选模型壳单元。如果其他部件附着在壳体上，则使用厚壳体有利于避免偏移。具有三维复杂应力状态的零件应采用solid单元进行建模。形状简单的零件优于六面体/五面体元件。把复杂的几何部分留给四面体单元。四面体单元类型4在当前版本中是最好的。四面体类型16和17使用10个节点可用。
- ◆ 避免使用10型四面体单元和2型六面体单元(全积分单元)。总的来说，四面体10型是一个很好的单元，与显示的测试结果形成对比。

内容

一、简介

二、ANSYS Drop Test Wizard

三、模型简化规则

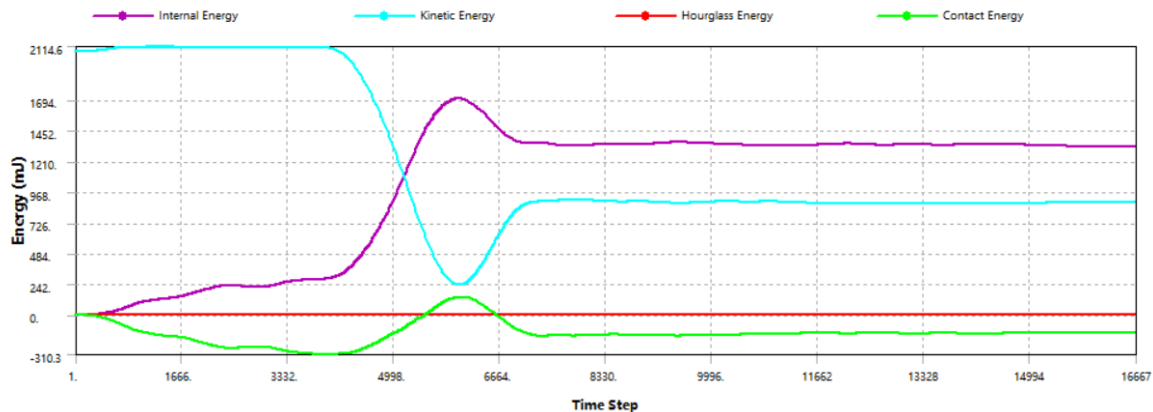
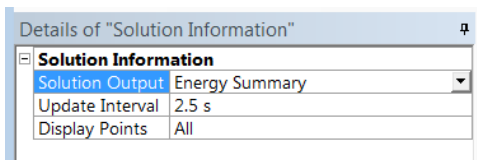
四、单元类型及选择

五、计算结果评估



5.1 能量平衡

用于检查模型是否正确。



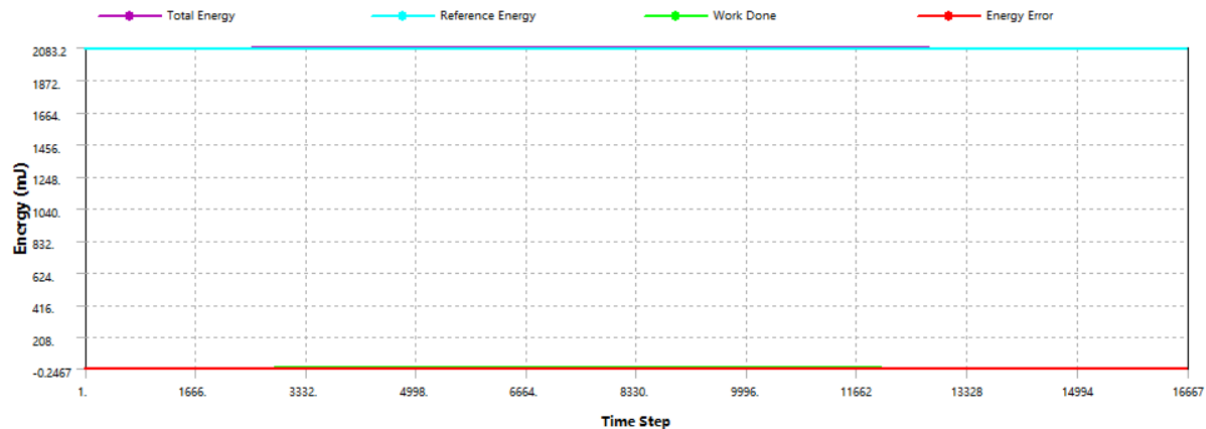
上图所示，动能与内能正确，但是接触能出现负值，表明模型存在原始穿透，需要检查模型网格，尽量保证模型没有穿透。

注：对于跌落模型，初始动能可用 $W = \frac{1}{2}mv^2$ 计算，保证模型能量和计算的能量一致。

5.1 能量平衡

用于检查模型是否正确。

| Details of "Solution Information" ⌵ | |
|--|------------------------------------|
| <input type="checkbox"/> Solution Information | |
| <input type="checkbox"/> Solution Output | Energy Conservation ▾ |
| Update Interval | 2.5 s |
| Display Points | All |



上图所示，总能量曲线是平滑曲线，与参考能量一致。

5.2 质量缩放

用于检查模型是否正确。

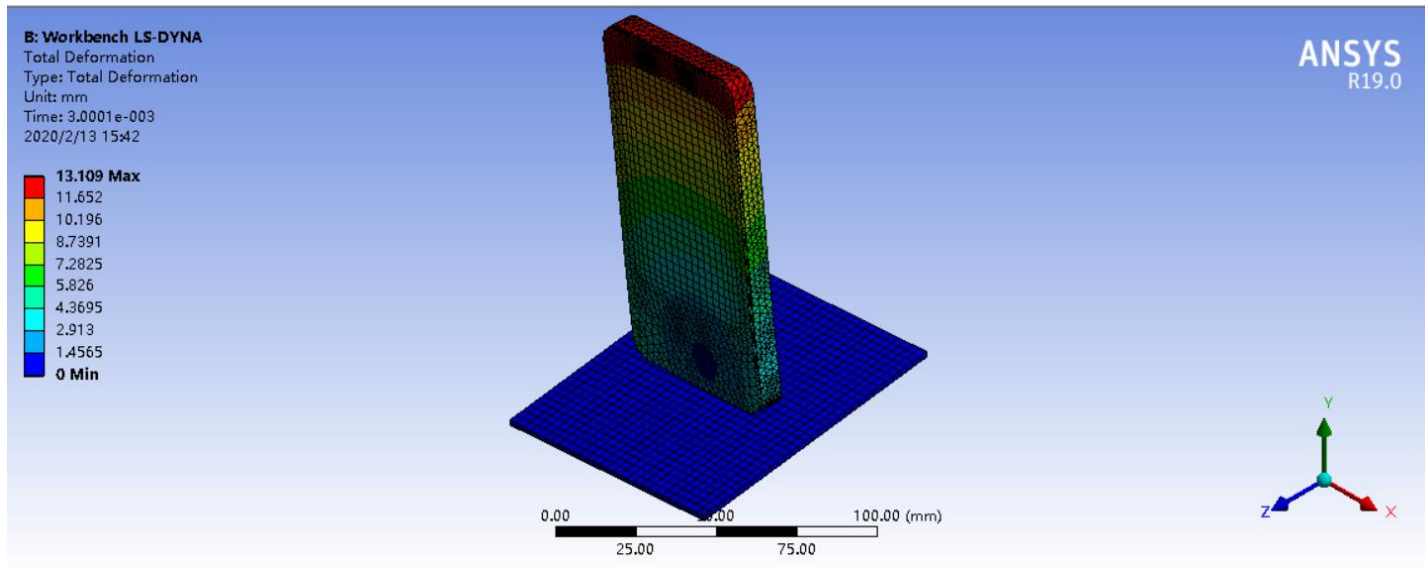
```
calculation with mass scaling for minimum dt
added mass = 1.1001E-06
physical mass= 2.7340E-04
ratio = 4.0237E-03
1 t 0.0000E+00 dt 1.80E-07 flush i/o buffers 02/13/20 14:50:53
1 t 0.0000E+00 dt 1.80E-07 write d3plot file 02/13/20 14:50:53
1 t 0.0000E+00 dt 1.80E-07 write intfor file 02/13/20 14:50:53
cpu time per zone cycle..... 0 nanoseconds
average cpu time per zone cycle.... 0 nanoseconds
average clock time per zone cycle.. 484 nanoseconds

estimated total cpu time = 2 sec ( 0 hrs 0 mins)
estimated cpu time to complete = 0 sec ( 0 hrs 0 mins)
estimated total clock time = 245 sec ( 0 hrs 4 mins)
estimated clock time to complete = 243 sec ( 0 hrs 4 mins)

added mass = 1.1001E-06
percentage increase = 4.0237E-01
```

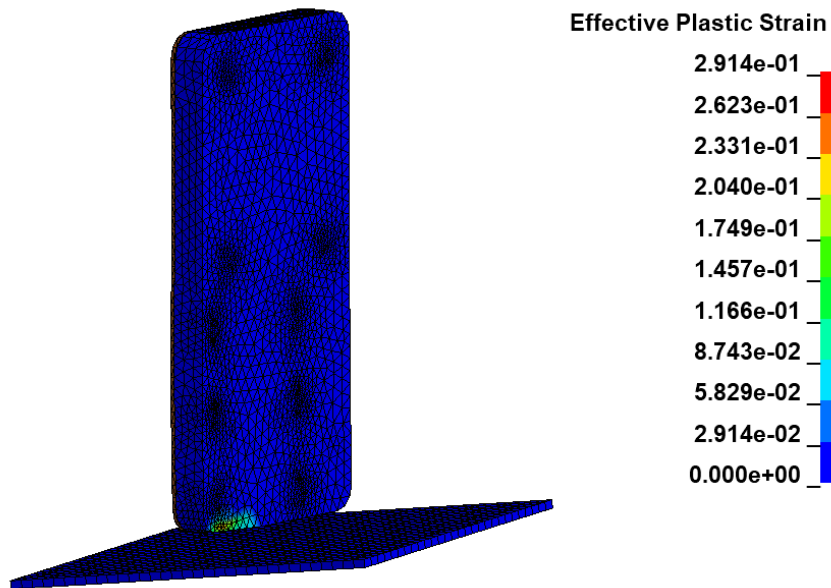
5.3 模型整体变形

用于检查模型是否正确。



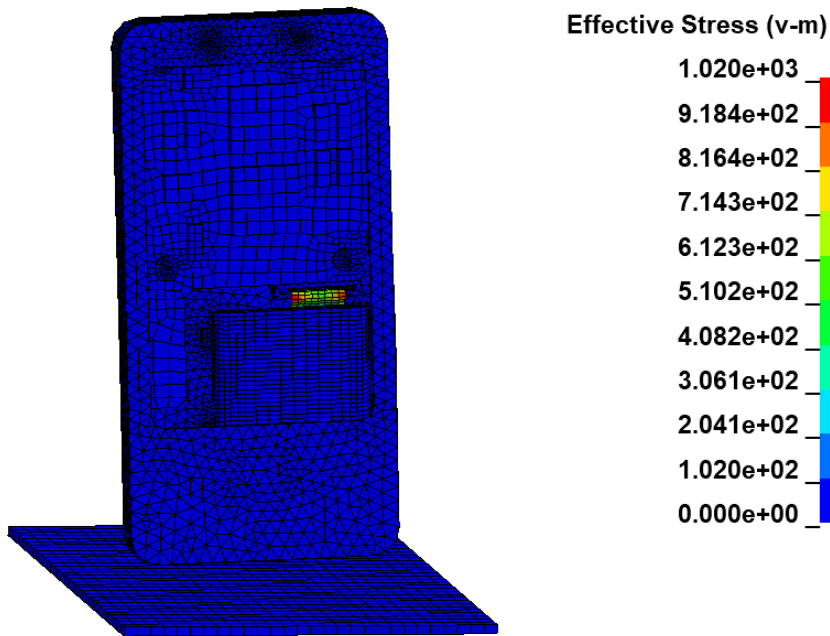
5.5 模型应变图

用于检查模型是否出现较大塑性应变以及判定部件失效。



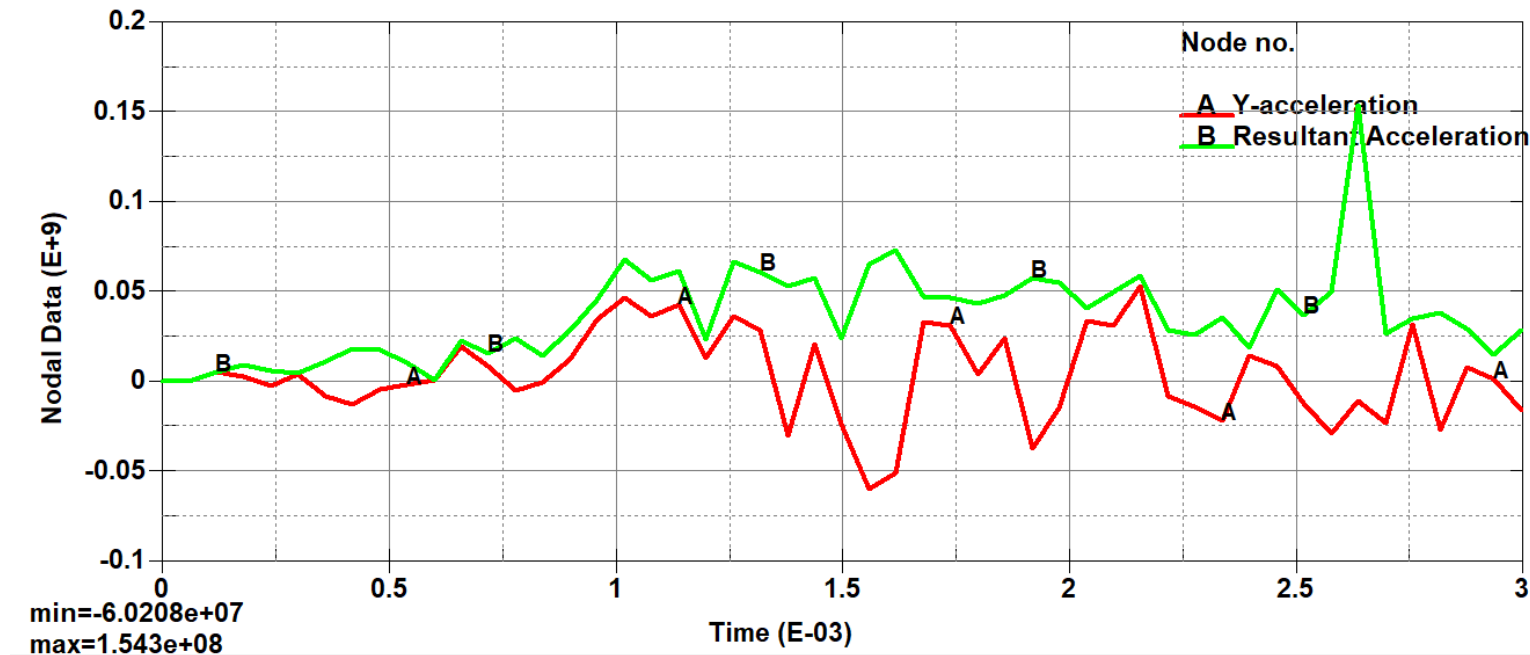
5.5 模型应力图

用于检查模型是否出现较大塑性应变以及判定部件失效。



5.5 模型加速度图

用于检查模型是否出现较大加速度值以及判定部件失效。





大咖慧，顾名思义，汇集众多大咖智慧。

是由安世亚太打造的一个以设计、仿真、增材制造等领域技术和行业专家为主的智慧学习平台。目前主要通过线上培训、研讨等方式，由行业相关领域资深专家与学员们分享交流最新技术和应用研究成果。

如有任何需求、建议，请关注订阅号（peraglobal），给我们留言

