



## **Lecture 1**

# **Overview of ABAQUS/Explicit**

Copyright 2005 ABAQUS, Inc.

## **Overview of ABAQUS/Explicit**

- What is Explicit Dynamics?
- ABAQUS/Explicit vs. ABAQUS/Standard
- Some Challenging Problems
- Defining an ABAQUS/Explicit Procedure
- Stable Time Increment
- Bulk Viscosity Damping
- Energy Balance
- Monitoring Diagnostic Messages
- Output



Copyright 2005 ABAQUS, Inc.





## What is Explicit Dynamics?

Copyright 2005 ABAQUS, Inc.

### What is Explicit Dynamics?

- **Dynamic equilibrium**

- The dynamic equilibrium equations are written for convenience with the inertial forces isolated from the other forces:

$$M\ddot{u} = P - I$$

- These equilibrium equations are completely general.
  - They apply to the behavior of *any* mechanical system and contain all nonlinearities (large deformations, nonlinear material response, contact).
  - When the first term—the inertial or dynamic force—is small enough, the equations reduce to the static form of equilibrium.

## What is Explicit Dynamics?

- **Explicit dynamics is a mathematical technique for integrating the equations of motion through time.**

- The explicit dynamic integration method is also known as the forward Euler or central difference algorithm.
  - Unknown values are obtained from information already known.
- Combining the explicit dynamic integration rule with elements that use a lumped mass matrix is what makes an explicit finite element program work.
- The lumped mass matrix,  $M$ , allows the program to calculate the nodal accelerations easily at any given time,  $t$ , using the following expression:

$$\ddot{u}\big|_{(t)} = M^{-1} \cdot (P - I)\big|_{(t)},$$

where  $P$  is the external load vector and  $I$  is the internal load vector.



## ABAQUS/Explicit vs. ABAQUS/Standard

## ABAQUS/Explicit vs. ABAQUS/Standard



- ABAQUS/Standard and ABAQUS/Explicit are intended to provide the user with two complementary analysis tools.
- ABAQUS/Standard provides the capability to analyze the following types of problems:
  - Linear and nonlinear static
  - Linear dynamic
  - Low-speed (low frequency response) nonlinear dynamic
  - Nonlinear heat transfer
  - Coupled temperature-displacement (quasi-static)
  - Coupled thermal-electrical
  - Mass diffusion problems
  - Structural-acoustics

## ABAQUS/Explicit vs. ABAQUS/Standard



- ABAQUS/Explicit provides the capability to analyze the following types of problems:
  - High-speed (short duration) dynamics
    - Drop tests and crash analyses of structural members
  - Large, nonlinear, quasi-static analyses
    - Deep drawing, blow molding, and assembly simulations
  - Highly discontinuous postbuckling and collapse simulations
  - Coupled temperature-displacement (dynamic)
    - Discussed in the *Heat Transfer and Thermal-Stress Analysis with ABAQUS* lecture notes
  - Structural-acoustics
    - Discussed in the *Structural-Acoustic Analysis with ABAQUS* lecture notes

## ABAQUS/Explicit vs. ABAQUS/Standard



- Explicit: Unknown values are obtained from information already known.
  - Neither iteration nor convergence checking is required.
  - The time increment has to be small enough in order to lie on the curve.
- Implicit: Unknown values (at the current time) are obtained from the current information.
  - Iteration and convergence checking are required.
  - The out-of-balance force is used to check equilibrium; the equation has to be solved over and over again...very time consuming!
  - Once convergence is achieved, however, the time increment can be large.

## ABAQUS/Explicit vs. ABAQUS/Standard



- **Advantages of ABAQUS/Standard**
  - The advantages of ABAQUS/Standard are:
    - It can solve for true static equilibrium,  $\mathbf{P} - \mathbf{I} = \mathbf{0}$ , in structural simulations.
    - It provides a large number of element types for modeling many different types of problems.
    - It provides analysis capabilities for studying a wide variety of nonstructural problems.
    - It uses a very robust and proven contact algorithm.
    - It uses an integration method for transient problems that has no mathematical limit (stability limit) on the size of the time increment—the time increment size is limited only by the desired accuracy of the solution.

## ABAQUS/Explicit vs. ABAQUS/Standard



- **Advantages of ABAQUS/Explicit**

- The advantages of ABAQUS/Explicit are:
  - It has been designed to solve highly discontinuous, high-speed dynamic problems efficiently.
  - It has a very robust contact algorithm that does not add additional degrees of freedom to the model.
  - It does not require as much disk space as ABAQUS/Standard for large problems, and it often provides a more efficient solution for very large problems.
  - It contains many capabilities that make it easy to simulate quasi-static problems.

## ABAQUS/Explicit vs. ABAQUS/Standard



- **When should ABAQUS/Explicit be used?**

- The deciding factor when choosing between ABAQUS/Standard and ABAQUS/Explicit is often the smoothness of the solution.
  - It may not be possible to obtain an efficient solution with ABAQUS/Standard if there are significant discontinuities in the solution.
- Possible sources of discontinuity in a solution:
  - Impact
  - Buckling or local wrinkling of the material
  - Material degradation or failure, such as cracking of concrete
- A large three-dimensional model that contains one or more of the discontinuities listed above is a good candidate for ABAQUS/Explicit.

## ABAQUS/Explicit vs. ABAQUS/Standard



- **Why use ABAQUS/Explicit?**

- **It expands that range of problems you can address.**

- ABAQUS/Explicit contains many modeling capabilities that do not exist in ABAQUS/Standard.
      - For example, material failure with element deletion for elastic-plastic materials
    - It can simulate larger models more readily with a given amount of computer hardware.

- **It is easy to learn.**

- The basic input structure and options for an ABAQUS/Explicit model are the same as those for an ABAQUS/Standard model.

- **It is economical to add ABAQUS/Explicit to your network.**

- The network licensing system allows a site to get both analysis programs at an attractive cost.



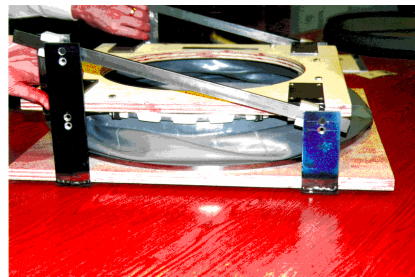
## Some Challenging Problems

## Some Challenging Problems

- We motivate further discussion of ABAQUS/Explicit by briefly reviewing some complicated problems suitable for analysis with ABAQUS/Explicit.
  - Rubber door seal
  - Wire crimping
  - Gas tank impact
  - Column buckling
  - Metal forming
  - Wiper blade

## Some Challenging Problems

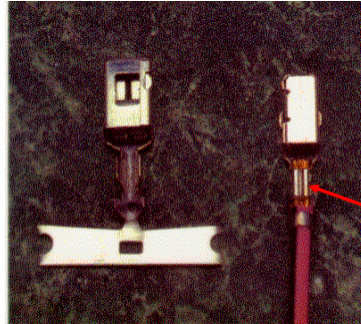
- Rubber door seal
  - This is an example of a typical door seal in washer machines.
  - Note the wrinkles shown in the picture.
    - The wrinkles will cause fatigue and premature failure.
    - ABAQUS/Standard cannot handle this type of analysis.



## Some Challenging Problems

### • Wire crimping

- You will find over 2000 pieces like these in an automobile.
- The pieces are designed to survive over the life span of the automobile.
- Critical design parameters include
  - Crimping height and width
  - Crimping force and pull out force
- Sophisticated detailed analysis is required.
  - Extensive and complicated contact conditions.

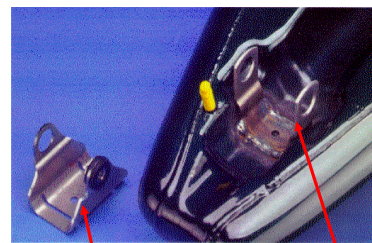


Crimp joint

## Some Challenging Problems

### • Gas tank impact

- Impact of any gas tank (automotive, locomotive) is similar to this one.
- The new rear bracket is wider than the original rear bracket, and the welding is inside of the bracket instead of outside for the original one.
- This decision is made based on analysis before the full-bike prototype test.
- Analysis significantly reduces project time.



New bracket

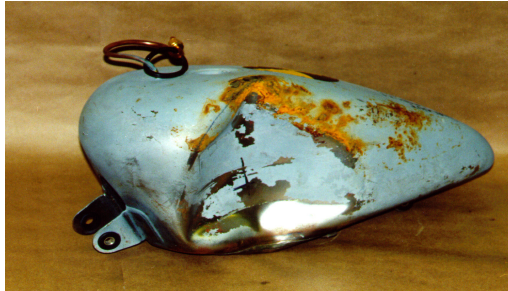
Original bracket

Motorcycle gas tank

## Some Challenging Problems



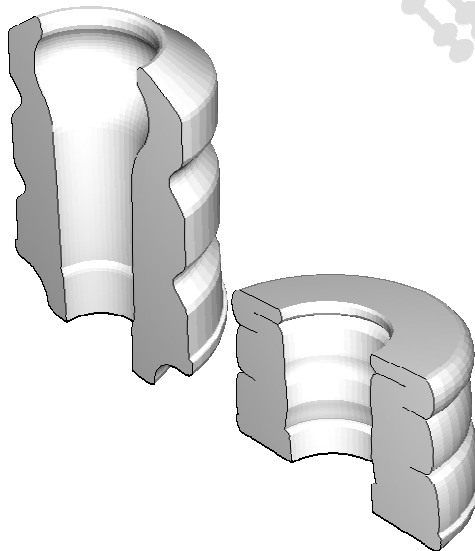
- **Gas tank impact (continued)**
  - Note the final permanent deformation after the impact



## Some Challenging Problems



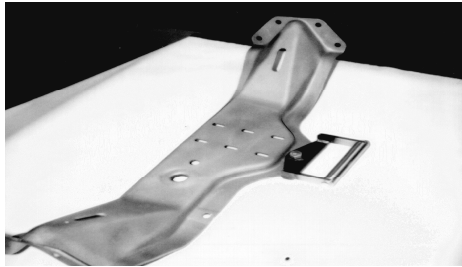
- **Column buckling**
  - Any buckling of a column structure involves self-contact.
  - Example: Jounce bumper
    - Uses very compressible material (soft rubber).
  - Analyses can be 2-D, axisymmetric, or fully 3-D.



## Some Challenging Problems

- **Metal forming**

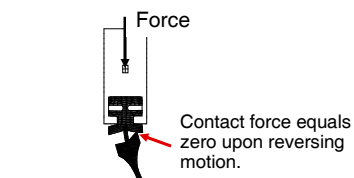
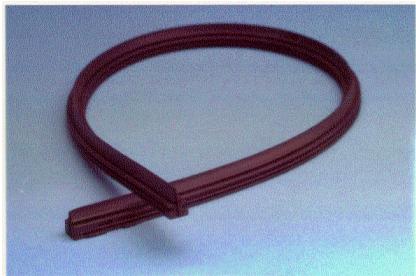
- ABAQUS/Explicit is used to simulate complicated forming processes
- Many contact constraints and friction effects
- ABAQUS/Standard is **less practical** to solve this class of problem



## Some Challenging Problems

- **Wiper blade**

- This is a very simple structure, but ABAQUS/Standard will not proceed past the dynamic transition point without employing special stabilization techniques.
  - This is the point when there is no vertical force between the ribs of the wiper blade, and this will destabilize the wiper blade when it is in motion.





## Defining an ABAQUS/Explicit Procedure

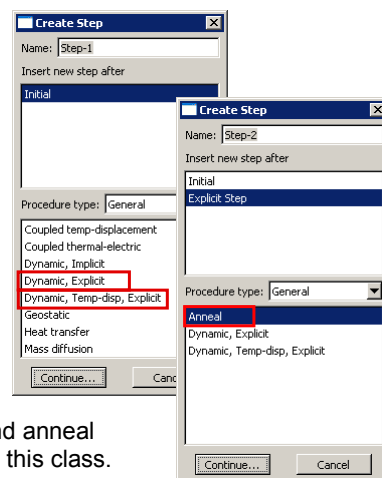
Copyright 2005 ABAQUS, Inc.

### Defining an ABAQUS/Explicit Procedure

- An ABAQUS/Explicit analysis is performed when the model contains any of the following procedure options:

- \*DYNAMIC, EXPLICIT  
an explicit dynamics step
- \*DYNAMIC TEMPERATURE-DISPLACEMENT, EXPLICIT  
a coupled thermo-mechanical explicit dynamics step
- \*ANNEAL:  
an anneal step; all nodal velocities are set to zero and all state variables, such as stress and plastic strain, are set to zero.

The coupled temperature-displacement and anneal procedures will not be discussed further in this class.



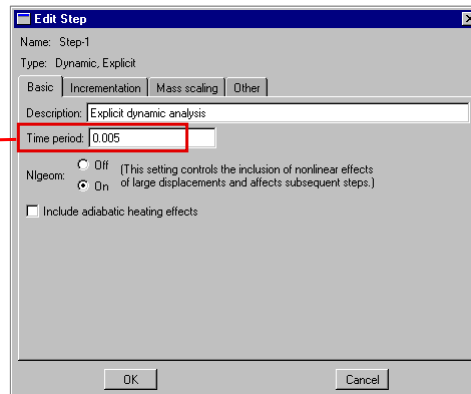
Copyright 2005 ABAQUS, Inc.



## Defining an ABAQUS/Explicit Procedure

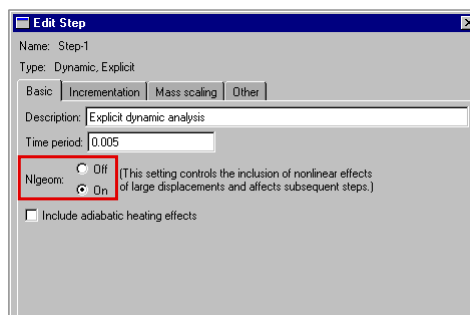
- For the majority of ABAQUS/Explicit analyses only the total step time needs to be specified.
  - Time increment size is chosen automatically so that it always satisfies the stability limit.

\*STEP  
\*DYNAMIC, EXPLICIT  
    ,  $t_{total}$



## Defining an ABAQUS/Explicit Procedure

- ABAQUS/Explicit uses a finite-strain, large-displacement, large-rotation formulation by default.
  - The NLGEOM parameter is not needed on the \*STEP option.
  - Geometrically linear analysis (small deformation analysis) can be obtained by including the NLGEOM=NO parameter.





## Explicit Stable Time Increment

Copyright 2005 ABAQUS, Inc.

### Stable Time Increment

- **The explicit dynamics procedure solves every problem as a wave propagation problem.**
  - Out-of-balance forces are propagated as stress waves between neighboring elements.
  - A bounded solution is obtained only when the time increment ( $\Delta t$ ) is less than the stable time increment ( $\Delta t_{\min}$ ).
    - If  $\Delta t \geq \Delta t_{\min}$ , the solution will be unstable and oscillations will occur in the model's response.
  - The stability limit can be defined in terms of the highest eigenvalue in the model ( $\omega_{\max}$ ) and the fraction of critical damping ( $\xi$ ) in the highest mode:

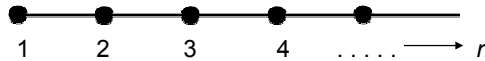
$$\Delta t_{\min} \leq \frac{2}{\omega_{\max}} \left( \sqrt{1 + \xi^2} - \xi \right).$$

- Thus, damping reduces the stable time increment.

## Stable Time Increment

- **Dilatational wave speed and the stable time increment**

- The concept of a stable time increment is explained easily by considering a one-dimensional problem:



One-dimensional problem

- The stable time increment is the minimum time that a dilatational (i.e., pressure) wave takes to move across any element in the model.
- Dilatation consists of volume expansion and contraction.

## Stable Time Increment

- The dilatational wave speed,  $c_d$ , can be expressed for a linear elastic material (with Poisson's ratio equal to zero) as

$$c_d = \sqrt{\frac{E}{\rho}},$$

where

- $E$  is the Young's modulus and  $\rho$  is the current material density.
- Based on the current geometry each element in the model has a characteristic length,  $L^e$ .
  - For shells and membranes the stability limit is based on the midplane or membrane dimensions only; for beams it is based on the axial dimensions.
  - When the transverse shear stiffness is defined directly for shell elements, the stable time increment will also consider the transverse shear behavior.

## Stable Time Increment

– Thus, the stable time increment can be expressed as

$$\Delta t = \frac{L^e}{c_d}$$

– Decreasing  $L^e$  and/or increasing  $c_d$  will reduce the size of the stable time increment.

- Decreasing element dimensions reduces  $L^e$ .
- Increasing material stiffness increases  $c_d$ .
- Decreasing material compressibility increases  $c_d$ .
- Decreasing material density increases  $c_d$ .

## Stable Time Increment

### • Example: Compression of a jounce bumper

– ABAQUS/Explicit writes a report to the status (.sta) file during the **datacheck** phase of the analysis.

- The initial stable time increments listed do not include damping (bulk viscosity) or mass scaling effects.

Initial time increment = 5.09466E-04

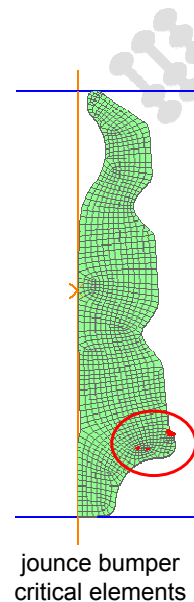
Statistics for all elements:

Mean = 1.53743E-03

Standard deviation = 3.20027E-04

Most critical elements :

Element number (Instance name)	Rank	Time increment	Increment ratio
485	1	5.09466E-04	1.00000E+00
PART-1-1			
230	2	5.21312E-04	9.77277E-01
PART-1-1			
224	3	5.21978E-04	9.76030E-01
PART-1-1			
⋮			
185	10	5.68758E-04	8.95751E-01
PART-1-1			

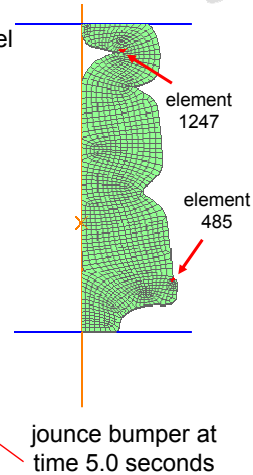


## Stable Time Increment

### • Example (cont'd): Compression of a jounce bumper

- In nonlinear problems the highest frequency of the model continuously changes; therefore, so does the stability limit.
- In problems with large deformations, element distortion may result in a stable time increment reduction.

STEP	TOTAL		STABLE	CRITICAL	KINETIC
INCREMENT	TIME	...	INCREMENT	ELEMENT	ENERGY
...					
5892	3.000E+00	...	5.087E-04	485	2.457E+02
6876	3.500E+00	...	5.074E-04	485	2.730E+02
7861	4.000E+00	...	5.070E-04	485	3.581E+02
8876	4.500E+00	...	3.120E-04	1247	4.106E+02
10702	5.000E+00	...	2.739E-04	1247	4.208E+02
12890	5.500E+00	...	2.065E-04	1247	3.859E+02



## Stable Time Increment

### • ABAQUS/Explicit has two types of time incrementation.

- Automatic time incrementation
  - ABAQUS/Explicit automatically adjusts the stable time increment during the analysis based on one of two methods:
    - Global estimation (default)
    - Element-by-element estimation
- Fixed time incrementation
  - A constant time increment is used for the entire analysis.
  - The fixed time increment can be
    - the initial element-by-element stable time increment, or
    - a specified time increment value.

## Stable Time Increment



- **Element-by-element stable time increment estimate**
  - The element-by-element estimate is determined using the current dilatational wave speed in each element.
  - The element-by-element estimate is conservative;
    - it will give a smaller stable time increment than the true stability limit that is based on the maximum frequency of the entire model.

## Stable Time Increment



- **Global stable time increment estimation**
  - An analysis starts by using the element-by-element estimation method.
  - The switch to the global estimation method occurs once the algorithm determines that the accuracy of the global estimation method is acceptable.
  - The adaptive, global estimation algorithm determines the maximum frequency of the entire model using the current dilatational wave speed.
    - This algorithm continuously updates the estimate for the maximum frequency.
    - The global estimator will usually allow time increments that exceed the element-by-element values.
  - The global estimation method is used by default for most analyses.
    - Some features prevent the use of the global estimator, such as material damping, distortion control, and adaptive meshing.

## Stable Time Increment

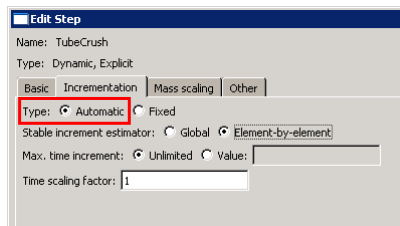
- For most analysis the default automatic global estimation method is appropriate.
  - Occasionally, experienced users may choose an alternative time incrementation method.
    - For example, to use a more conservative stable time increment.
- The calculated stable time increment values can be scaled.
  - This is available for all but the specified fixed time increment method.

## Stable Time Increment

- Specifying a non-default time incrementation method.
  - The incrementation method options are part of the explicit dynamic procedure definition:

**\*DYNAMIC, EXPLICIT, [ELEMENT BY ELEMENT | FIXED TIME INCREMENTATION | DIRECT USER CONTROL], [SCALE FACTOR = value]**

$t_{total} \dots \Delta t_{max}$



**Edit Step**

Name: TubeCrush

Type: Dynamic, Explicit

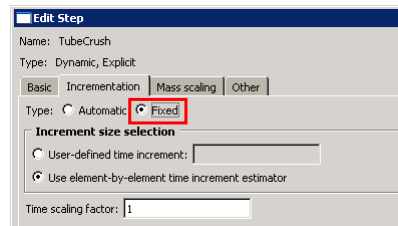
Basic Incrementation Mass scaling Other

Type: ☒ Automatic ☐ Fixed

Stable increment estimator: ☐ Global ☒ Element-by-element

Max. time increment: ☒ Unlimited ☐ Value:

Time scaling factor:



**Edit Step**

Name: TubeCrush

Type: Dynamic, Explicit

Basic Incrementation Mass scaling Other

Type: ☐ Automatic ☒ Fixed

Increment size selection

☐ User-defined time increment:

☒ Use element-by-element time increment estimator

Time scaling factor:

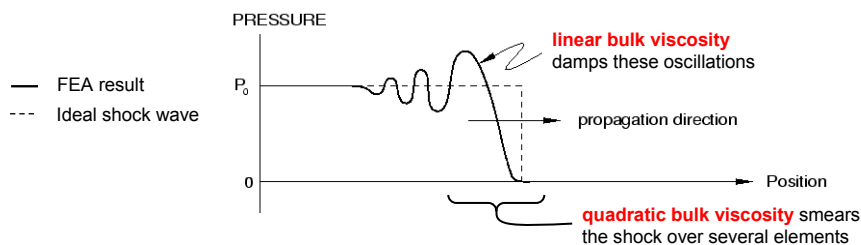


## Bulk Viscosity Damping

Copyright 2005 ABAQUS, Inc.

### Bulk Viscosity Damping

- Bulk viscosity introduces damping associated with volumetric straining.
  - Its purpose is to improve the modeling of high-speed dynamic events.
- There are two forms of bulk viscosity: linear bulk viscosity and quadratic bulk viscosity.
  - The effects of the bulk viscosity forms on a shock wave propagating through a one dimensional mesh are shown below.



propagation of a shock wave through a one-dimensional mesh.

Copyright 2005 ABAQUS, Inc.



## Bulk Viscosity Damping

- A small amount of bulk viscosity is included by default for all ABAQUS/Explicit analysis.
  - The default value of the linear damping coefficient  $b_1$  is 0.06.
  - The default value of the quadratic damping coefficient  $b_2$  is 1.2.
- The default bulk viscosity is appropriate for most models; however, the damping coefficients can be modified if necessary.

- To modify bulk viscosity for the entire model:

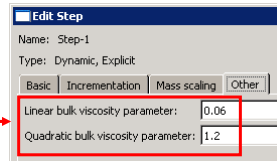
**\*BULK VISCOSITY**  
 $\langle b_1 \rangle, \langle b_2 \rangle$

- To modify bulk viscosity for an element set:

**\*SECTION CONTROLS, NAME=name**  
 $\langle b_1 \text{ scaling factor} \rangle, \langle b_2 \text{ scaling factor} \rangle$

and refer to the section controls by name on the element section definition, for example:

**\*SOLID SECTION, ELSET=PAD, MATERIAL=FOAM, CONTROLS=NoDamping**



## Energy Balance

## Energy Balance

### • Energy balances

- An energy balance equation can be used to help evaluate whether an ABAQUS/Explicit simulation is yielding an appropriate response.
- In ABAQUS/Explicit the energy balance equation is written as

$$E_I + E_{VD} + E_{FD} + E_{KE} - E_W = E_{TOT} = \text{constant},$$

where

- $E_I$  is the internal energy (elastic, inelastic, “artificial” strain energy),
- $E_{VD}$  is the energy absorbed by viscous dissipation,
- $E_{FD}$  is the frictional dissipation energy,
- $E_{KE}$  is the kinetic energy,
- $E_W$  is the work of external forces, and
- $E_{TOT}$  is the total energy in the system.

## Energy Balance

- These energies can be requested for the entire model or for subsets of the model.
- Indicators of problems when reviewing the energy balance are:
  - Excessive “artificial” strain energy that is used to suppress hourglass modes.
    - It should be less than 1–2% of  $E_I$  for all but the enhanced hourglass control method.
    - Hourglassing is discussed in detail in Lecture 2, *Elements*.
  - Excessive kinetic energy in a quasi-static simulation.
    - The kinetic energy should be a small fraction (typically 5–10%) of  $E_W$  or  $E_I$ .
    - Remember that rigid bodies with mass contribute to the total kinetic energy in the model.
    - Quasi-static analysis is discussed in detail in Lecture 5, *Quasi-Static Analyses*.

## Energy Balance



- Indicators of problems (cont'd):
  - A large change in the total energy of the model.
    - Such a change could be caused by exceeding the stability limit ( $\Delta t > \Delta t_{\min}$ )
    - The presence of conflicting constraints can also result in a change in total energy.
      - There is an example of this in Lecture 4, *Contact Modeling*.



## Monitoring Diagnostic Messages

## Monitoring Diagnostic Messages



- **There are several diagnostic warning messages that may be issued by ABAQUS/Explicit during an analysis:**
  - Deformation speed warnings are issued when an element is deforming so fast that it is in danger of collapsing.
  - Warped surface warnings indicate that the contact surfaces are so distorted that contact tracking may fail.
  - Shell element thickness warnings indicate that the thickness of the shell elements may be large enough, relative to the membrane dimensions, to cause problems in contact.
  - Initial overclosure warnings are issued when significant overclosures exist and may produce erroneous solutions.

## Monitoring Diagnostic Messages



- **Warnings and errors are printed to the status (.sta) file.**
  - Additional diagnostic details may be printed to the message (.msg) file.
- **The \*DIAGNOSTICS option controls diagnostic messaging.**
  - By default, short summaries of diagnostic checks are written to the status or message files if problems are detected.
  - Use the \*DIAGNOSTICS option to request detailed diagnostic output.
  - The \*DIAGNOSTICS option is not currently supported by ABAQUS/CAE.
    - It can be included using the Keywords Editor.



## Output

Copyright 2005 ABAQUS, Inc.

## Output

- **Four types of output are available:**

- Neutral binary output can be written to the output database (.odb) file.
- Restart output can be written to the restart (.res) file.
  - This output is used for:
    - conducting restart analyses
      - discussed in Lecture 10, *Managing Large Models*
    - transferring results between ABAQUS/Explicit and ABAQUS/Standard
      - discussed in Lecture 11, ABAQUS/Explicit–ABAQUS/Standard interface
- Selected results (.se1) file output can be written for third-party postprocessing.
  - It is not discussed further here.

Copyright 2005 ABAQUS, Inc.



## Output

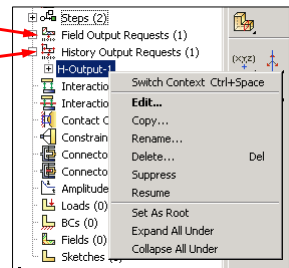
### • Output to the output database file

- The output database file is used by ABAQUS/Viewer.
  - An interface (API) is available in Python and C++ to use for external postprocessing (e.g., to add data to display in ABAQUS/Viewer).
- Two types of output data: *field* and *history* data.
  - Field data are used for model (deformed, contour, etc.) and X–Y plots.

\*OUTPUT, FIELD

- History data are used for X–Y plots.

\*OUTPUT, HISTORY



## Output

- Example of a history output request:

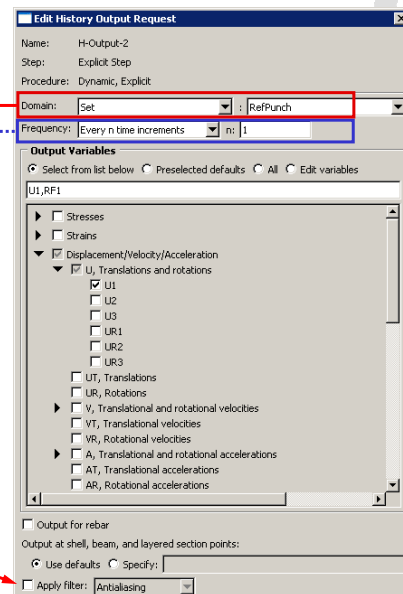
Output can be restricted to specific regions of the model.

The frequency at which output is written can be controlled.

\*OUTPUT, HISTORY, FREQUENCY=1

\*NODE OUTPUT, NSET=RefPunch  
U1, RF1

Real-time filtering of history data is available. See Lecture 10, *Managing Large Models* for more information.



## Output

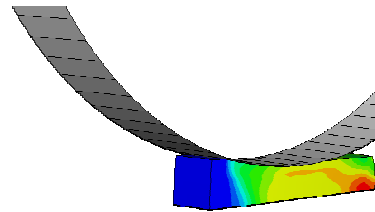
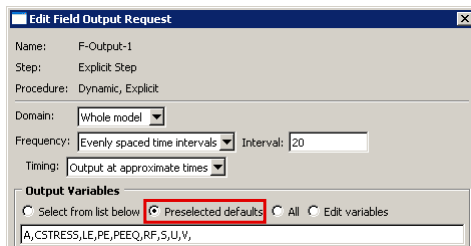
– Preselected output includes the most commonly used output variables for a given analysis procedure.

- The following field output variables are saved by default:

– Element variables: S, LE, PE, PEEQ

– Nodal and surface variables: U, V, A, RF, CSTRESS

**\*OUTPUT, FIELD, VARIABLE=PRESELECT**



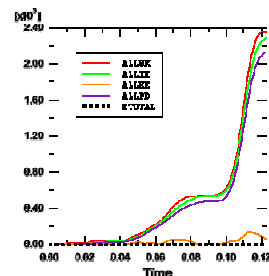
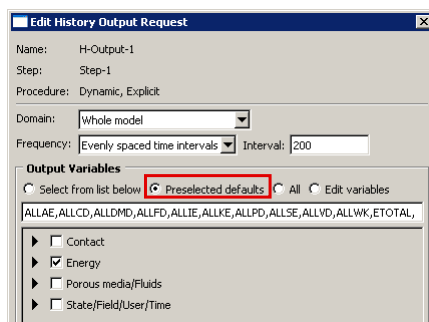
Rolling analysis deformed shape with contours of equivalent plastic strain (PEEQ)

## Output

- The default history output consists of all the global (whole model) energies:

– ALLKE, ALLSE, ALLWK, ALLPD, ALLCD, ALLVD, ALLDMD, ALLAE, ALLIE, ALLFD, ETOTAL

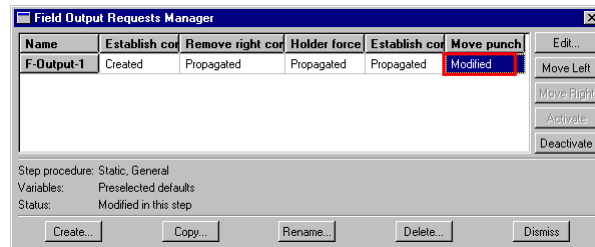
**\*OUTPUT, HISTORY, VARIABLE=PRESELECT**



Energy history for a wire crimping analysis

## Output

- Output requests are propagated to subsequent steps defined in the model.
- The request can be modified in a later step.



**\*OUTPUT, FIELD, OP=[NEW | REPLACE | ADD]**