Lecture 11

ABAQUS/Explicit–ABAQUS/Standard Interface

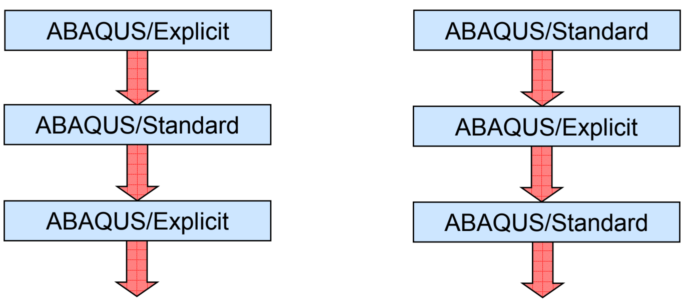
Overview

- Introduction
- Import from ABAQUS/Explicit to ABAQUS/Standard
- Import from ABAQUS/Standard to ABAQUS/Explicit
- Additional Import Modeling Issues
- Limitations
- Demonstration

Introduction

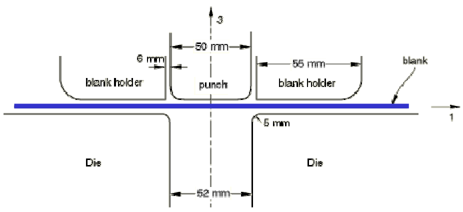
Introduction

- **ABAQUS provides a capability to transfer a deformed mesh and associated state between an ABAQUS/Explicit analysis and an ABAQUS/Standard analysis.**

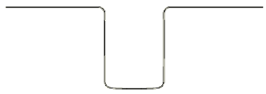


Introduction

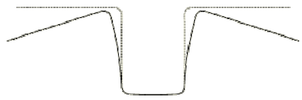
– The deformed model can be transferred **from ABAQUS/Explicit to ABAQUS/Standard** to:
  - Obtain the final static configuration after a dynamic event
  - Simulate springback
  - Perform eigenvalue or buckling simulations on the formed part



Initial assembly for forming process



Blank after forming (ABAQUS/Explicit)

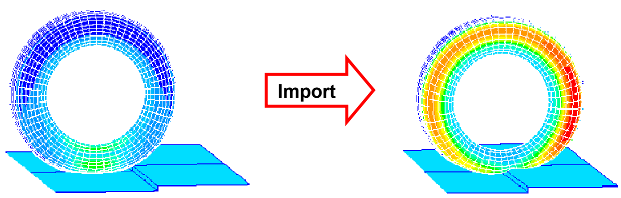


Springback (ABAQUS/Standard)

Two-dimensional draw bending forming and springback analysis

Introduction

– The deformed model can be transferred **from ABAQUS/Standard to ABAQUS/Explicit** to:
  - Simulate a preloading phase where the capabilities of ABAQUS/Explicit are not needed (no contact, etc.).
  - Simulate further forming steps after an intermediary springback phase.
  - Simulate a transient event after steady-state conditions have been found (e.g., with a steady-state transport simulation).



Tire footprint and steady-state rolling (ABAQUS/Standard)

Tire hitting curb (ABAQUS/Explicit)

Introduction

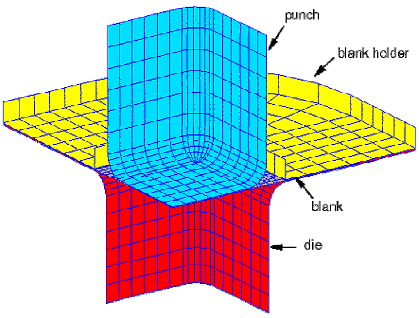
- **Summary of import capabilities:**

Can be imported	Need to be respecified	Cannot be imported
Material state	Boundary conditions	Analytical rigid surfaces
Nodal positions	Loads	Mesh-independent fasteners
Elements, element sets	Contact definitions	Connector elements
Nodes, node sets	Output requests	Dashpot and spring elements
Temperatures	Kinematic constraints	Mass and rotary inertia elements
Rebar layers	Nodal transformations	Infinite elements
	Amplitude definitions	Fluid elements

Import from ABAQUS/Explicit to ABAQUS/Standard

Import from ABAQUS/Explicit to ABAQUS/Standard

- **The following example demonstrates the procedure for transferring results from ABAQUS/Explicit to ABAQUS/Standard.**
- **Example: Deep drawing of a square box**
  - The blank undergoes large deformations during the sheet metal forming process.
  - Once the forming process is complete and the confining tools are removed, the blank will attempt to recover its elastic deformation.
  - This *springback* phenomenon may lead to unacceptable warping of the formed product.
    - Forming tools must be designed to compensate for springback effects.



Initial mesh of deformable blank and rigid tools. One-quarter of the box is modeled.

Import from ABAQUS/Explicit to ABAQUS/Standard

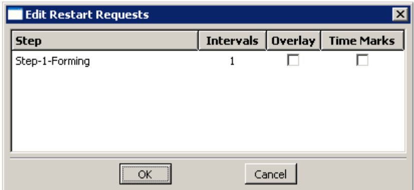
- For the calculation of springback associated with the sheet metal forming processes:

1. The forming process is simulated using ABAQUS/Explicit.
2. The deformed mesh of the blank and its associated state at the end of the forming process are imported into ABAQUS/Standard.
3. The springback calculation is performed in ABAQUS/Standard.
   - The displacement field that ABAQUS/Standard calculates is the amount of springback that occurs provided that the reference configuration is updated.
     - If it is not, the displacements will be total values, including the forming phases.

Import from ABAQUS/Explicit to ABAQUS/Standard

1 The forming process is simulated using ABAQUS/Explicit.

– During the analysis, restart data must be written at the time when transfer of the model's state is required (i.e., at the end of the forming analysis).

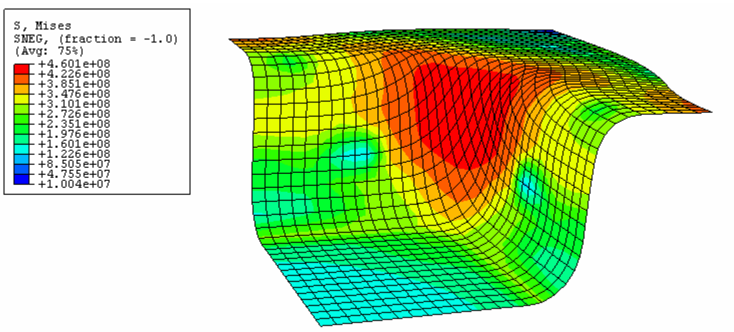



```
*RESTART, WRITE, NUMBER INTERVAL=1
```

– The ABAQUS/Explicit analysis will produce the following files which will be used to import the final deformed shape of the blank into ABAQUS/Standard.
  - State (`.abq`),
  - Analysis database, (`.stt`, and `.pac`),
  - Part (`.prt`), and
  - Output database (`.odb`) files.

Import from ABAQUS/Explicit to ABAQUS/Standard

– Results at the end of the ABAQUS/Explicit forming analysis:



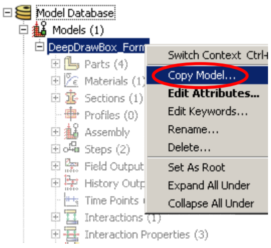
Mises stress contours for the bottom surface of the blank

Import from ABAQUS/Explicit to ABAQUS/Standard

2 **The deformed mesh of the blank and its associated state at the end of the forming process are imported into ABAQUS/Standard.**

ABAQUS/CAE interface

a Copy the forming model into a new model.



Keywords interface

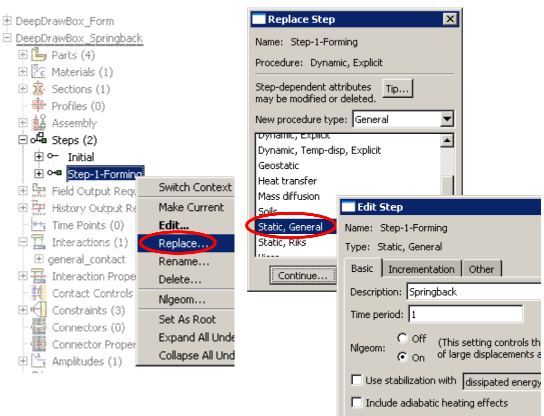
a In a text editor and begin a new input file.

```
*HEADING
Springback Calculation
```


Import from ABAQUS/Explicit to ABAQUS/Standard

ABAQUS/CAE interface

b Replace (and rename) the dynamic ABAQUS/Explicit step with a static ABAQUS/Standard step.



Keywords interface

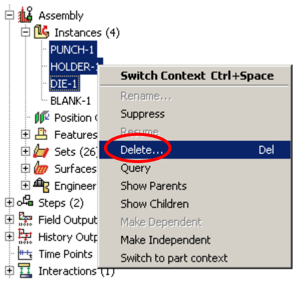
b Define an ABAQUS/Standard static step.

```
*HEADING
Springback Calculation
*IMPORT, STEP=1, INTERVAL=1,
  UPDATE=YES
 BLANK,
**
*STEP,NLGEOM
*STATIC
 0.1,1.
*BOUNDARY
B1,YSYMM
B4,XSYMM
*BOUNDARY, TYPE=VELOCITY
CORNER, 3, 3, 0.0
*END STEP
```

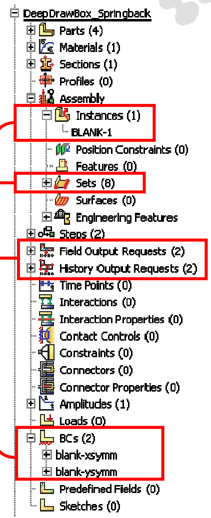

Import from ABAQUS/Explicit to ABAQUS/Standard

ABAQUS/CAE interface only

c1 Delete interactions, rigid tools and associated sets, boundary conditions, etc.



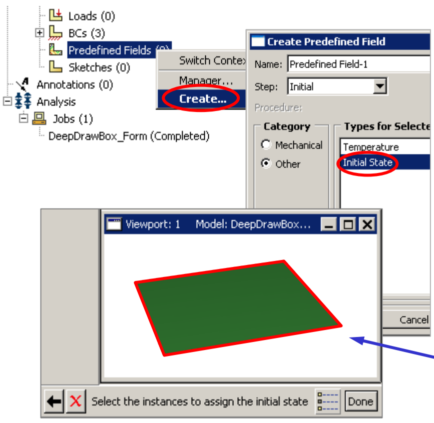
c2 Keep blank and associated sets, boundary conditions, etc.



Import from ABAQUS/Explicit to ABAQUS/Standard

ABAQUS/CAE interface

d1 Define an initial state field to import forming analysis results for the blank part instance.



Keywords interface

d1 Import element set **BLANK** from the forming analysis.

```
*HEADING
Springback Calculation
*IMPORT, STEP=1, INTERVAL=1,
  UPDATE=YES
BLANK,
**
```

BLANK was the element set used for the blank section definition in the original forming analysis:

`*SHELL SECTION, ELSET=BLANK`

a requirement for input files not defined as assemblies of part instances.

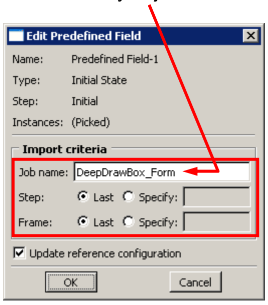
In ABAQUS/CAE and input files defined using parts and assemblies, part instances are specified for import.

Import from ABAQUS/Explicit to ABAQUS/Standard

ABAQUS/CAE interface

d2 Import the initial state field from the end of the ABAQUS/Explicit forming analysis.

Name of the forming analysis job.



Keywords interface

d2 Import the blank from the end of the ABAQUS/Explicit forming analysis.

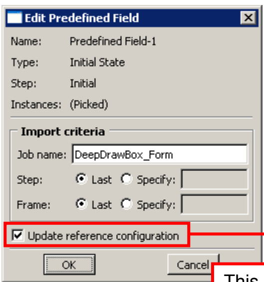
If the **STEP** and **INTERVAL** parameters are omitted, the data would be imported from the last available step and interval for which restart data were saved

```
*HEADING
Springback Calculation
*IMPORT, STEP=1, INTERVAL=1,
  UPDATE=YES
 BLANK,
**
*STEP,NLGEOM
*STATIC
 0.1,1.
*BOUNDARY
B1,YSYMM
B4,XSYMM
*BOUNDARY, TYPE=VELOCITY
CORNER, 3, 3, 0.0
*END STEP
```


Import from ABAQUS/Explicit to ABAQUS/Standard

ABAQUS/CAE interface

d3 Update the reference configuration



Keywords interface

d3 Update the reference configuration

```
*HEADING
Springback Calculation
*IMPORT, STEP=1, INTERVAL=1,
  UPDATE=YES
 BLANK,
**
*STEP,NLGEOM
*STATIC
 0.1,1.
       RY
       M
       M
       RY, TYPE=VELOCITY
       , 3, 3, 0.0
*END STEP
```

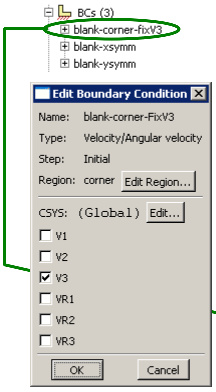
This option is used so that the reference configuration is the deformed model prior to importing.

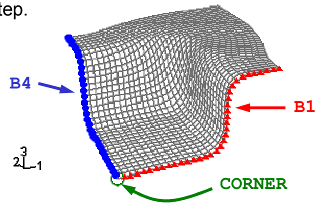
The displacement field that ABAQUS/Standard calculates will be the amount of springback that occurs.

Import from ABAQUS/Explicit to ABAQUS/Standard

ABAQUS/CAE interface

e Define a boundary condition to prevent rigid body motion of the blank in the springback step.





Imported node sets

Keywords interface

e Define boundary conditions for the springback step.

```
*HEADING
Springback Calculation
*IMPORT, STEP=1, INTERVAL=1,
  UPDATE=YES
 BLANK,
**
*STEP,NLGEOM
*STATIC
 0.1,1.
*BOUNDARY
B1,YSYMM
B4,XSYMM
*BOUNDARY, TYPE=VELOCITY
CORNER, 3, 3, 0.0
*END STEP
```

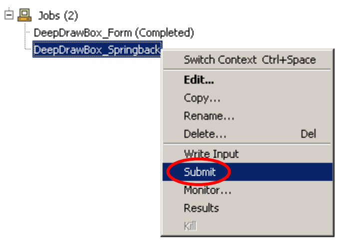
Respecify symmetry boundary conditions

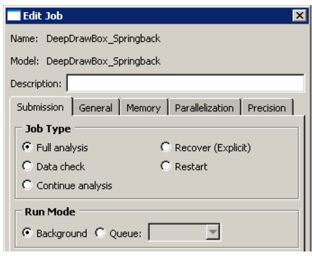
 ABAQUS

Import from ABAQUS/Explicit to ABAQUS/Standard

3 The springback calculation is performed in ABAQUS/Standard.

– ABAQUS/CAE interface job submission:





– Keyword interface job submission:

- At the command prompt

```
abaqus job=DeepDrawBox_Springback oldjob=DeepDrawBox_Form
```

Name of ABAQUS/Standard springback analysis input file

Name of ABAQUS/Explicit forming analysis job

 ABAQUS

Import from ABAQUS/Explicit to ABAQUS/Standard

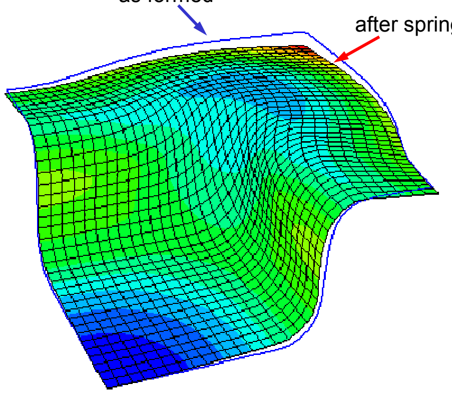
- **Equilibrium**
  - After importing the deformed blank and its current material state into ABAQUS/Standard, the model will not be in static equilibrium.
    - Two sets of forces contribute to this condition:
      - **Dynamic forces:**
        - The forming process is simulated using a dynamic procedure, so inertia and damping forces are present.
        - In a proper forming process simulation the state of dynamic equilibrium is relatively close to a state of static equilibrium.
      - **Boundary interaction forces:**
        - Changes in the boundary and contact conditions from the ABAQUS/Explicit analysis to the ABAQUS/Standard analysis contribute to the lack of static equilibrium.
        - Since springback involves the removal of contact conditions, this is the dominant contributing factor.

Import from ABAQUS/Explicit to ABAQUS/Standard

- The initial out-of-balance forces are defined as those needed to achieve static equilibrium:
  - When the deformed blank is imported with the material state into ABAQUS/Standard, a set of artificial internal stresses are applied that equilibrate the imported stresses so that static equilibrium is obtained at the start of the analysis.
  - These artificial stresses are removed gradually during the springback calculation step.
  - As the stresses are removed, the blank deforms further as a result of redistribution of internal forces.
  - The final (sprungback) configuration is achieved after complete removal of the artificial stresses or initial out-of-balance forces.

Import from ABAQUS/Explicit to ABAQUS/Standard

– The configuration of the box forming analysis after springback is shown in the figure.

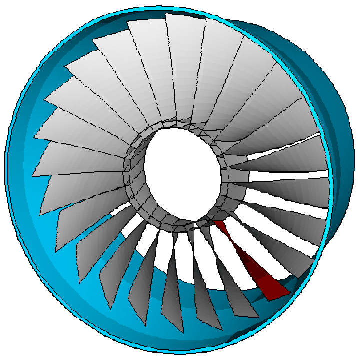


deformed configuration after springback
contours of displacement magnitude

Import from ABAQUS/Standard to ABAQUS/Explicit

Import from ABAQUS/Standard to ABAQUS/Explicit

- **The following example demonstrates the procedure for transferring results from ABAQUS/Standard to ABAQUS/Explicit.**
- **Example: Blade Containment**
  - A combination of ABAQUS/Standard and ABAQUS/Explicit is shown to be an effective tool for simulating a jet engine blade-out event.
  - The fan blade assembly is initially stationary.
    - We obtain the steady state spinning solution of the engine fan blade in ABAQUS/Standard
    - The results of this analysis are used as initial conditions in the dynamic blade release analysis in ABAQUS/Explicit.



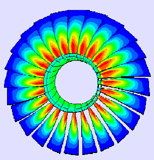
ABAQUS model of an aircraft engine; blade colored red is released

Import from ABAQUS/Standard to ABAQUS/Explicit

- **Modeling approach**
  - The results transfer capability is used to transfer the state associated with the steady state spinning solution from the implicit static procedure to the explicit dynamics procedure.

ABAQUS/Standard

Static procedure
Centrifugal Loading
⇩
Obtain associated stress state

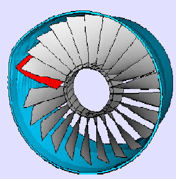


fan only

Import →

ABAQUS/Explicit

Import stress state
Initial Conditions: Spin at appropriate velocity
Verify Steady state
Throw Blade



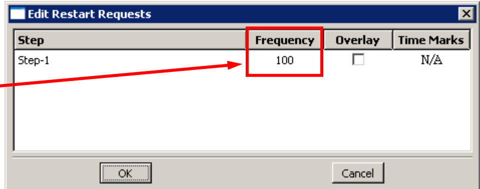
fan and casing

Import from ABAQUS/Standard to ABAQUS/Explicit

- **The input syntax for ABAQUS/Standard to ABAQUS/Explicit import is very similar to ABAQUS/Explicit to ABAQUS/Standard already discussed.**
  - Request restart data during the ABAQUS/Standard steady state spinning analysis:

```
*RESTART, WRITE, FREQUENCY=100
```

The frequency options is only available when requesting data for an ABAQUS/Standard analysis.



  - The ABAQUS/Standard analysis will produce the following files which will be used to import the spinning fan into ABAQUS/Explicit.
    - Restart (`.res`),
    - Analysis database, (`.stt`, and `.mdl`),
    - Part (`.prt`), and
    - Output database (`.odb`) files.

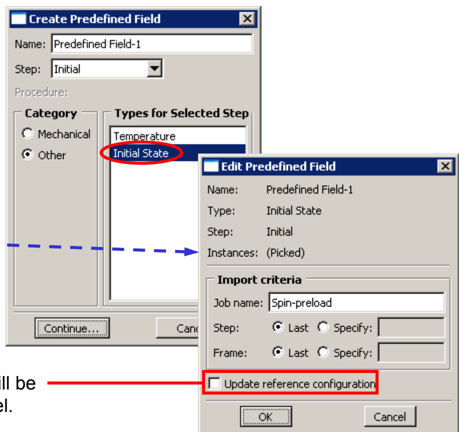
Import from ABAQUS/Standard to ABAQUS/Explicit

  - Import the ABAQUS/Standard steady state spinning results into the ABAQUS/Explicit blade throw model:

```
*IMPORT,STEP=1, UPDATE=NO
 HUB,
 BLADES,
```

Import the fan.

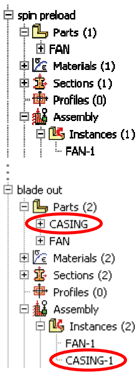
The sets **HUB** and **BLADES** were used on the section definition options for the hub and the blades.



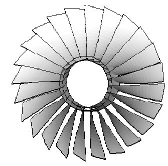
The reference configuration will be the original, undeformed model.

Import from ABAQUS/Standard to ABAQUS/Explicit

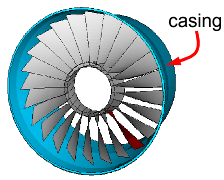
– The engine casing, which was not required in the ABAQUS/Standard steady state spinning analysis, is defined in the ABAQUS/Explicit blade throw model:




```
*IMPORT,STEP=1, UPDATE=NO
 HUB,
 BLADES,
**
** DEFINE CASING
*NODE
10001, 410, 90, 0
10002, 410, 300, 0
10003, 440, -100, 0
:
*ELEMENT, TYPE=S4R,ELSET=CASING
10001, 10305,10940,10953,10304
10002, 10940,10941,10954,10953
10003, 10941,10942,10955,10954
:
```



ABAQUS/Standard model



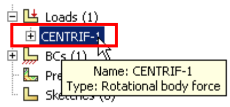
ABAQUS/Explicit model

 ABAQUS

Import from ABAQUS/Standard to ABAQUS/Explicit

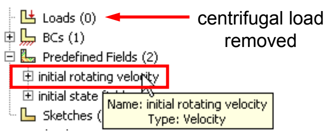
- **Loading**
  - In ABAQUS/Standard we apply a centrifugal load about a defined axis.

```
*DLOAD
** 3000 rpm --> 314.159 rad/sec --> 98696.044 (rad/s)^2
ROTOR, CENTRIF, 98696.044, 0., 0., 0., 0., 1., 0
```



  - In ABAQUS/Explicit, we apply an initial rotating velocity to the fan blades.

```
*INITIAL CONDITIONS, TYPE=ROTATING VELOCITY
ROTOR-N, 314.159, 0, 0, 0
0., 0., 0., 0., -1., 0
```



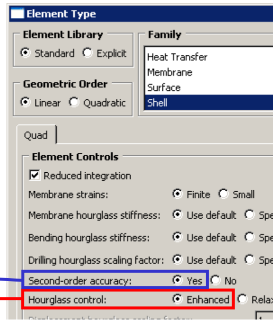
- Connector elements are used to attach a fan blade to the hub and subsequently release the blade.

 ABAQUS

Import from ABAQUS/Standard to ABAQUS/Explicit

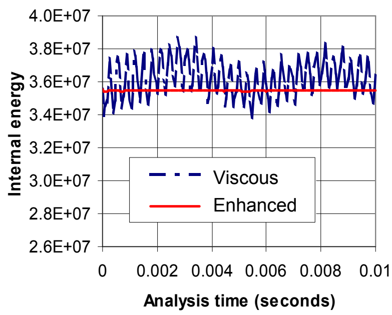
- **Element section controls**
  - Element section controls are defined in the original analysis and cannot be redefined in an import analysis.
    - Material definitions can be redefined upon import.
  - In this example, the following section controls are defined in the ABAQUS/Standard spinning analysis:
    - Enhanced hourglass control
      - Recommend for consistency between analyses
    - Second order accuracy element formulation
      - The default first order element formulation in ABAQUS/Explicit is not appropriate for a structure subjected to several revolutions.

```
*SECTION CONTROLS, NAME=ENHANCED,
  SECOND ORDER ACCURACY=YES,
  HOURGLASS=ENHANCED
```



Import from ABAQUS/Standard to ABAQUS/Explicit

- The enhanced hourglass control formulation is used so that hourglass forces are consistently computed in both analyses.
  - This leads to the reduction of noise in the import analysis solution.



ABAQUS/Explicit results for a steadily spinning disk (i.e., no blade release) after importing results from ABAQUS/Standard.

Import from ABAQUS/Standard to ABAQUS/Explicit

- **Other modeling issues**
  - **Use of general contact**
    - The large number of potential interactions is readily handled with general contact.
  - **Use of double precision executable**
    - Running in double precision is more suitable for this analysis because of the high number of revolutions of the fan blade assembly.
  - **Specifying a maximum stable time increment**
    - By specifying a maximum time increment, we guarantee that each degree of revolution of the fan blade is analyzed over at least 10 increments.
      - We recommend a minimum of 10 analysis increments per degree of revolution in problems involving finite rotations.

Import from ABAQUS/Standard to ABAQUS/Explicit

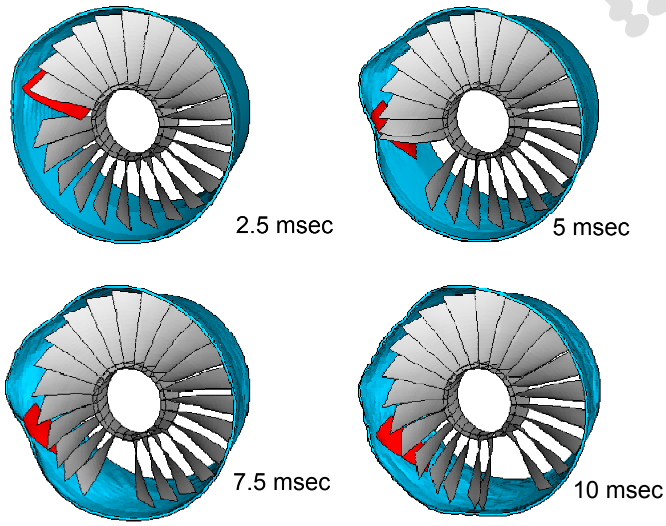
- **Results**

Many revolutions
(i.e. steady state verification)

Video Clip

Blade detachment

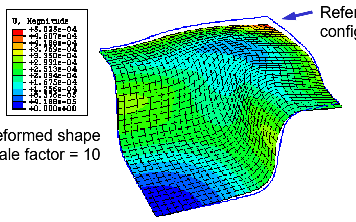
Video Clip



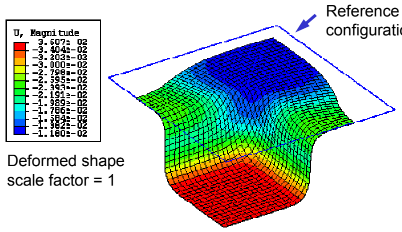
Additional Import Modeling Issues

Additional Import Modeling Issues

- **Reference configuration**
  - For springback simulations the reference configuration is usually updated.
    - The displacements, strains, step number, and total time are reset to zero for the current analysis.
  - For multi-stage forming simulations, do not update the reference configuration.
    - The displacements, strains, step number, and total time are continued from the previous analysis.



Reference configuration updated



Reference configuration not updated

Additional Import Modeling Issues

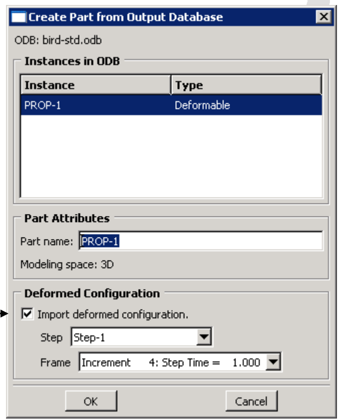
- **Importing the material state**
  - By default, the material state for each element is imported
    - This includes the stresses, the orientation of the stresses, and the following state variables:
      - equivalent plastic strains and for Mises plasticity
      - back stresses for the kinematic hardening models
      - the maximum strain energy density during deformation history for the Mullins effect
      - user-defined state variables for user-defined constitutive models (user subroutines **`VUMAT`** and **`UMAT`**).

Additional Import Modeling Issues

- If the material state is not needed, set the STATE parameter equal to NO on the *IMPORT option.

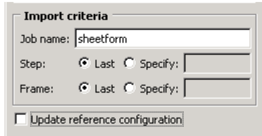
```
*IMPORT, STATE = NO
```

  - All state variables are reset to zero.
- ABAQUS/CAE **always** imports the material state along with the deformed mesh.
  - If you want to import only the deformed mesh, you can import an orphan mesh from a selected step and increment of an output database.

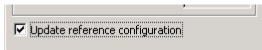


Additional Import Modeling Issues

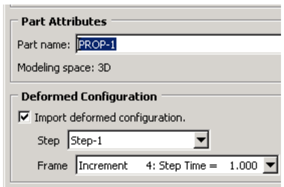
- **Reference configuration and material state combinations**
  - UPDATE=NO, STATE=YES:
    - Continue the previous analysis (most common usage)



  - UPDATE=YES, STATE=YES:
    - Current stress applied to the current configuration with no strain (similar to an initial stress in a regular analysis)



  - UPDATE=YES, STATE=NO:
    - Annealing



  - UPDATE=NO, STATE=NO:
    - (Awkward; have strains but no stresses)

Additional Import Modeling Issues

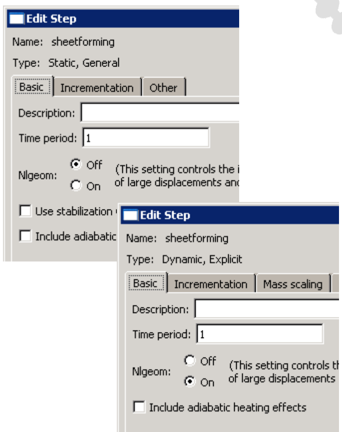
- **Material definitions**
  - The material property definitions are imported by default.
    - They can be redefined if necessary.
    - If redefined:
      - The material names used for the imported elements must be the same as the material names used in the previous analysis.
      - Caution should be exercised if changes are made to the material properties.
        - This may affect the initial out-of-balance forces.
      - The mass density may need to be redefined if mass scaling was used in an ABAQUS/Explicit simulation.
        - If mass scaling is used, the scaled masses will not be transferred to the subsequent import analysis.

Additional Import Modeling Issues

- **Element section definitions**
  - Element section property definitions are imported and may not be changed from those used in the previous analysis.
    - The current thickness of each shell element is imported automatically.
    - Any material orientation information is imported automatically when the material state is imported.
- **Valid elements for importing**
  - The import capability is available for first-order continuum, shell, membrane, beam (both linear and quadratic), truss, and rigid elements that are common to both ABAQUS/Explicit and ABAQUS/Standard.

Additional Import Modeling Issues

- **Geometric nonlinearity**
  - NLGEOM default settings:
    - NO for ABAQUS/Standard
    - YES for ABAQUS/Explicit
  - The setting of NLGEOM is imported and becomes the setting in the new analysis.
    - The reference configuration cannot be updated if NLGEOM=NO is used because, with small deformation, there is no reason to update the configuration.



Additional Import Modeling Issues

- **Initial conditions and boundary conditions**
  - Valid initial conditions for various values of the STATE parameter are shown below:

*INITIAL CONDITIONS, TYPE=	STATE=
HARDENING	NO
RELATIVE DENSITY	NO
ROTATING VELOCITY	YES or NO
SOLUTION	NO
STRESS	NO
VELOCITY	YES or NO
VOID RATIO	NO

- New boundary conditions must be specified.
  - These may be different from those used in the previous simulation.

ABAQUS

Limitations

Limitations

- The same versions of ABAQUS/Explicit and ABAQUS/Standard must be run on computers that are binary compatible.
- The capability is not available for spring, mass, dashpot, rotary inertia, connector, fluid, and infinite elements.
- Contact definitions cannot be transferred.
- Surface definitions (including analytical rigid surface definitions) cannot be transferred.
- Loads, boundary conditions, kinematic constraints (surface-based tie, coupling constraints, etc.), and nodal transformations are not imported.
- Mesh-independent spot welds are not imported.
    - However, the spot weld reference nodes are imported and can be used to redefine spot welds in the import analysis.
        - In this case, it is recommended that the reference configuration be updated if large deformations occurred in the imported model.

Limitations

- The complete material state (including state variables, such as plastic strains) is imported only for:
    - linear elasticity,
    - Mises plasticity (including the kinematic hardening models),
    - hyperfoam,
    - hyperelasticity,
    - Mullins effect,
    - and the user materials (VUMAT and UMAT).
- Material stresses are imported for all valid material models.

Limitations

– Analyses with predefined field variables and/or temperature fields cannot be imported.
  - Temperatures are imported for coupled temperature-displacement and adiabatic analyses if the material state is imported.
  - Coupled temperature-displacement elements with predefined field variable definitions can be imported, however the field variable values are not imported.

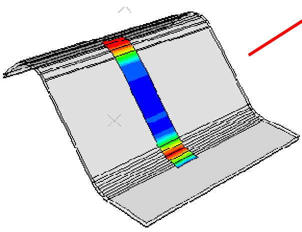
ABAQUS

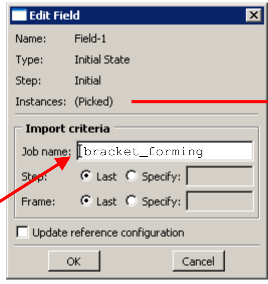
Demonstration

Demonstration

- **In this demonstration ABAQUS/CAE is used to model the forming and springback of an angled bracket.**

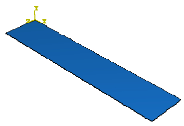
The forming of half of a symmetric angled bracket is analyzed using ABAQUS/Explicit





Springback is analyzed in ABAQUS/Standard.

An Initial state field is used to import the final deformed mesh and material state from the ABAQUS/Explicit forming analysis.



Play demo with audio

Play demo without audio