What's New in SIMULIA Structures Portfolio

Radek Linhart, Ondřej Marada, Ondřej Flídr

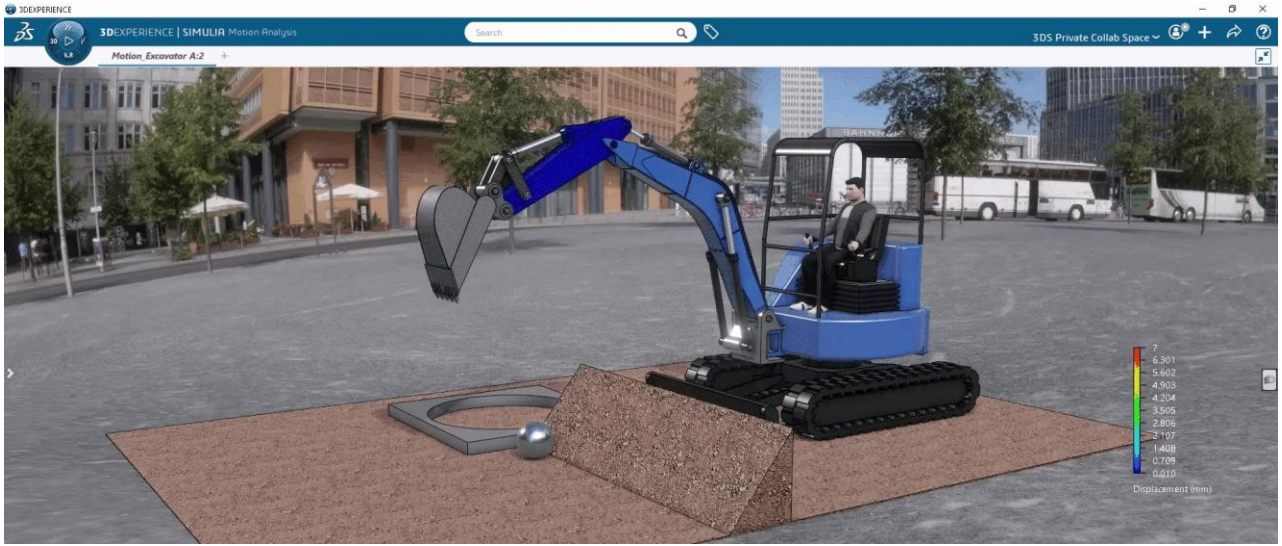
SIMULIA Days 2025

PUBLIC

Contents

2025 GA

- 3DEXPERIENCE platform now allows users to perform multibody simulations
- Full integration with 3DEXPERIENCE database – easy access to geometry, FEMs, Flexible bodies
- Event based multibody simulations
- Transfer multibody results to static analysis procedures

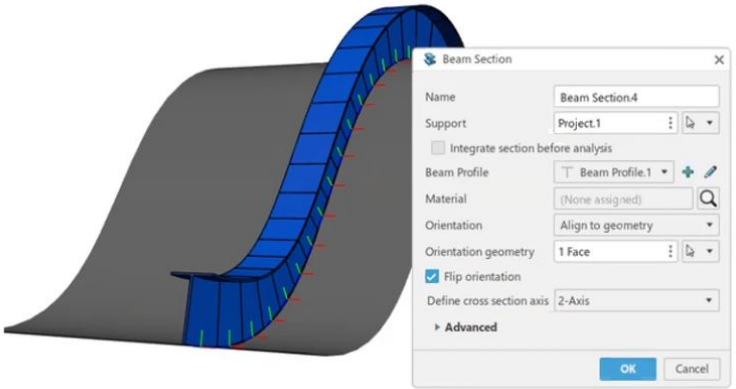


BEAM ORIENTATION

BASED ON CURVED SURFACES

2025 FD03

- Beam profile orientation can be based on a curved surface

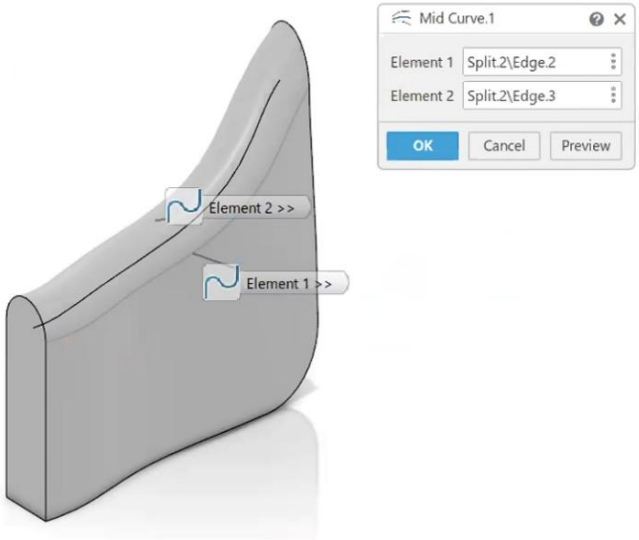


GEOMETRY PREPARATION
IMPROVEMENTS

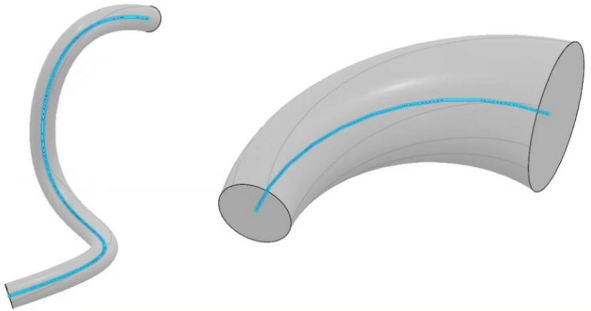
2026 GA

- Extract middle wire between 2 wires → manual ribs creation
- Extract neutral fiber from circular rod or pipe

Mid wire



Neutral fiber

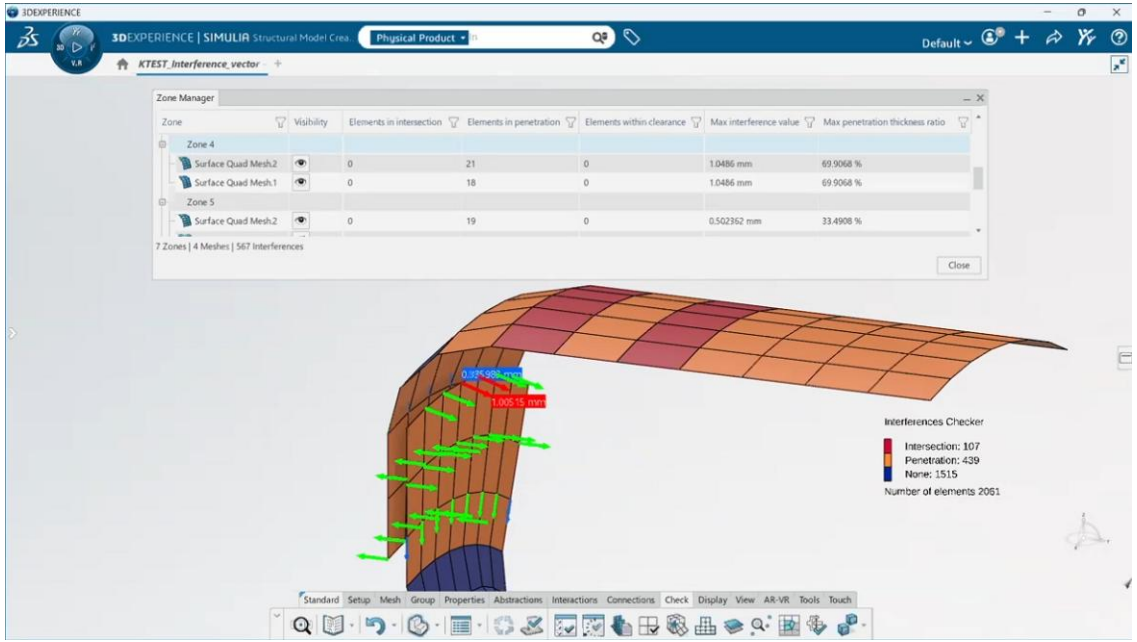


INTERFACE CHECKER

PENETRATION VECTORS AT NODES

2026 GA

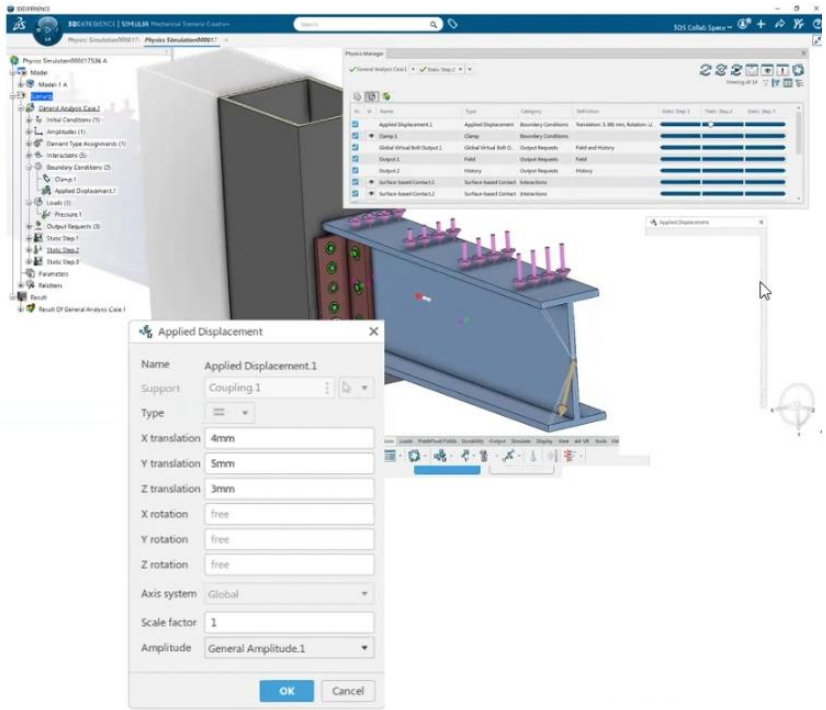
- Display of penetration vector at nodes
- Advanced thickness penetration review to support manual de-penetration



COMBINED FEATURES IN BC AND LOAD DEFINITIONS

2026 GA

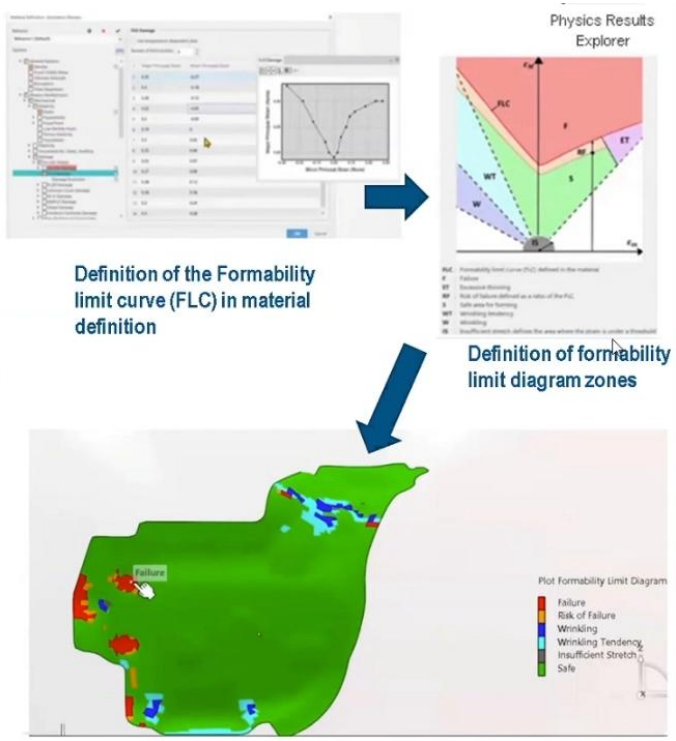
- New Boundary conditions
- New Loads
- Combines translational and rotational directions into one GUI



FORMABILITY LIMIT DIAGRAMS IN PHYSICS RESULTS EXPLORER

2026 GA

- Provided 3D color code plot for Formability Limit Diagram
- Based on Formability Limit Curve parameters from Material definition

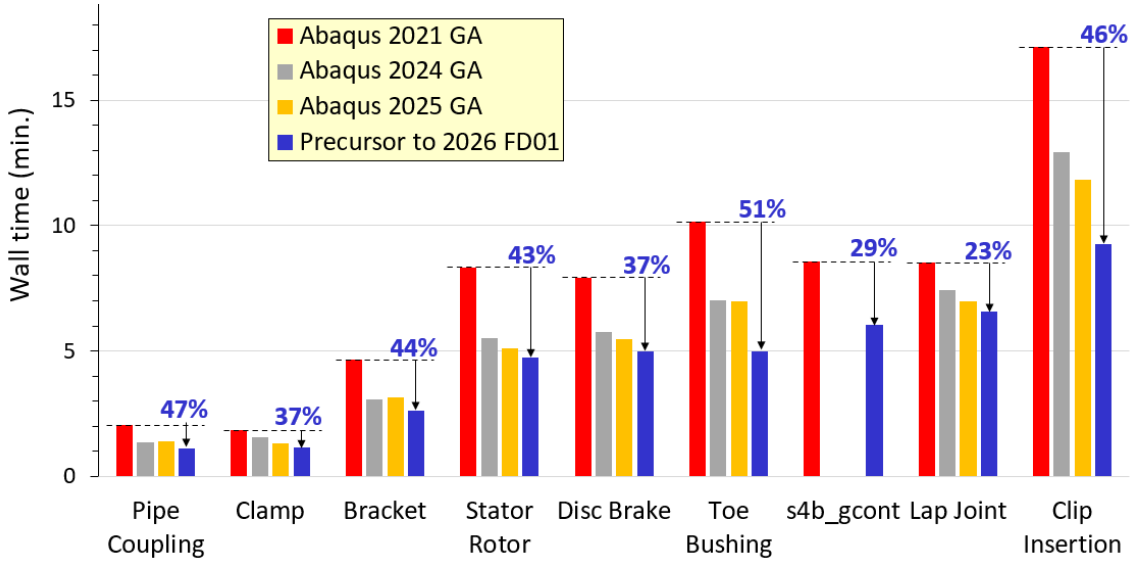


CONTENTS

Improved performace of General contact in Abaqus/Standard

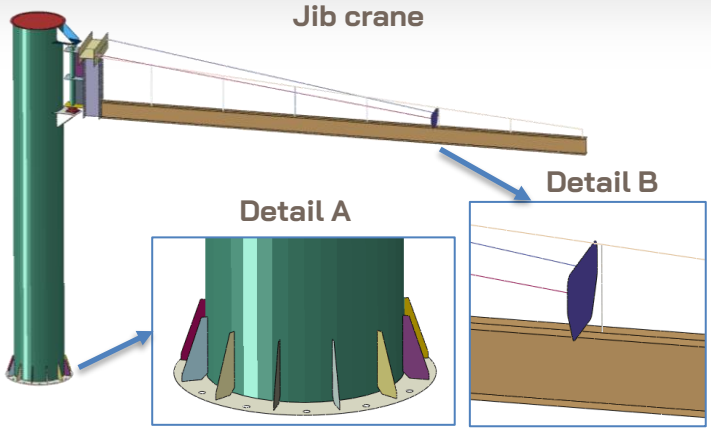
2026 FD01

- 12-30% faster in Abaqus 2026 FD01 vs Abaqus 2024 GA



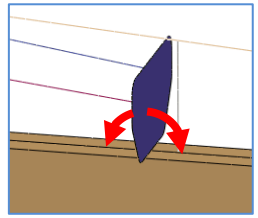
BONDING VIA COHESIVE CONTACT IN GENERAL CONTACT

MODIFICATION



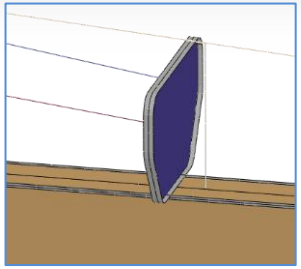
- **Surface-based tie constrain (*TIE)**
  - Necessary to define corresponding pairs, could be time consuming
  - Permanent bonds with infinite bond stiffness
  - *TIE will not constrain nodal rotations of a shell to a solid (Explicit)
    - ➔ Shell-to-solid coupling is an alternative

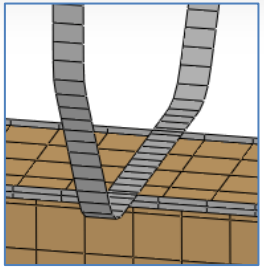
- **Alternative is cohesive contact in general contact**
  - Can be defined globally / locally
  - Permanent bonds with default cohesive stiffness, can be adjusted.
    (default cohesive stiffness for permanent cohesive bonds **10x higher in Abaqus/Std 2025 FD01**
    -> Makes results more consistent with *TIE)
  - Support degradation and eventual failure of the bond between two cohesive surfaces
  - An unconstrained “hinging” mechanism exists for cohesive contact at shell perimeters
    in past Abaqus versions



BONDING VIA COHESIVE CONTACT IN GENERAL CONTACT

MODIFICATION





- **Enhancement of bonding at shell perimeter**
  - Internally generate surface elements along shell perimeter during batch Pre **in Abaqus/Std 2025 FD01**
  - The added nodes are constrained to shell perimeter nodes
  - Relative rotations are resisted via contact force “couples” at surface-element nodes separated by shell thickness
    - ➔ Contact force couples at surface-elements nodes lead to moments at shell perimeter nodes

Analytic solution (u_y = 0,5120 mm)

Shell-to-solid coupling (u_y = 0,509 mm)

*Shell to solid coupling, constraint name=Constraint-1
shell_surf, solid_surf

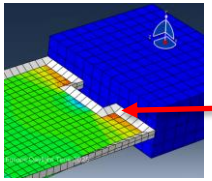
Cohesive contact (u_y = 0,5076 mm)

-> Default (very high) cohesive stiffness is in effect

2025 FD01

*Surface Interaction, Name=**PermBond**
*Cohesive Behavior, Cohere=Original Contacts
*Contact Initialization Data, Name=**Init**, Search Above=0.1

*Contact
*Contact Inclusions, All Exterior
*Contact Property Assignment
, , **PermBond**
*Contact Initialization Assignment
, , **Init**
*Contact Controls Assignment, **Shell Perimeter=Facet**



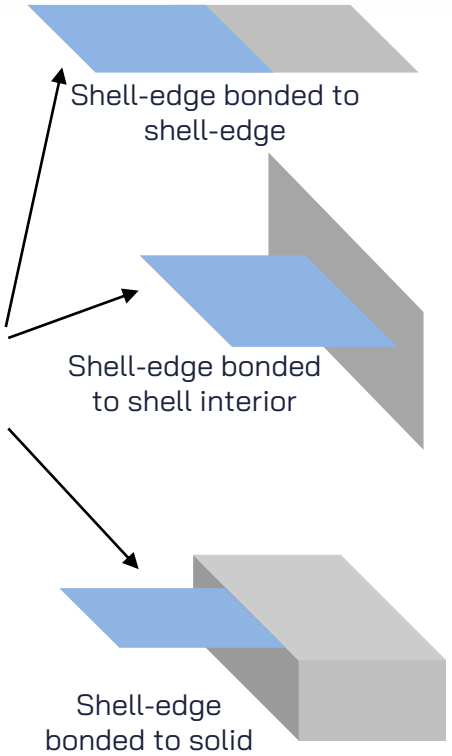
Visualization of internal surface element in Aba CAE

BONDING VIA COHESIVE CONTACT IN GENERAL CONTACT MODIFICATION

Reflects cohesive contact enhancements anticipated for Abaqus/Std 2025 FD01

Pre-existing advantages of cohesive contact vs. *TIE

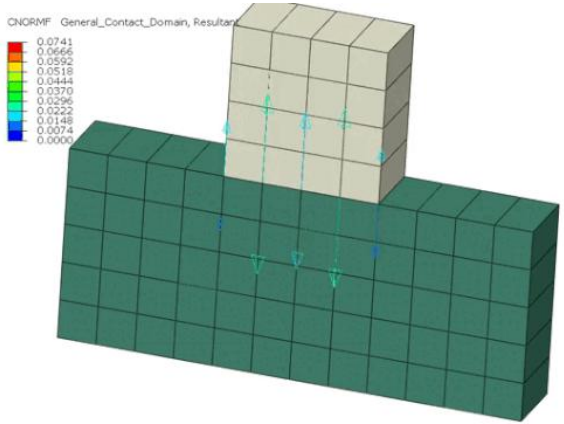
Desirable bond characteristics	*TIE	Cohesive contact A/Std 2025 FD01
1. Insignificant relative translation across bond	Yes	Almost always
2. Easily specify storing offsets for instead of nodal adjustments	Adjust=No (on different keyword options)	
3. Insignificant relative rotation if both surfaces have rot. DOF	Yes	Yes
4. Insignificant. relative rotation if only one surface has rot. DOF	No	
5. Avoid over-constraint issues		
6. Global assignment allowed		
7. Interface output variables		
8. Allow new bonds during sim.		
9. Bonds can damage and fail		



No Separation Contact in Abaqus/Explicit

2025 FD01

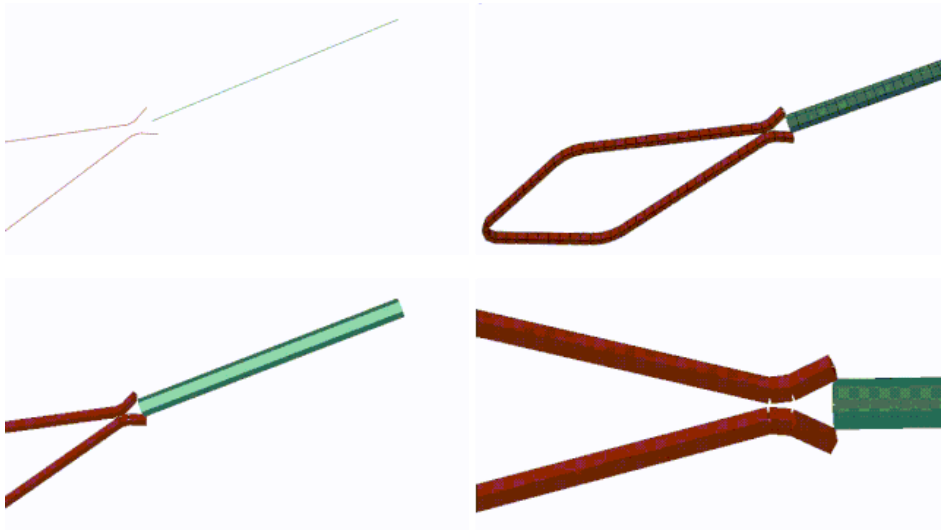
- When using general contact in Abaqus/Explicit, you can now define a no separation contact relationship without the need to use a rough frictional model.
- Previously, you could only specify no-separation behavior in combination with “rough” friction for General contact in Abaqus/Explicit
- “No separation” contact behavior in the normal direction does not prevent loss of contact `*SURFACE BEHAVIOR, NO SEPARATION`
- Benefits: You can solve new classes of problems in which surfaces in contact remain in contact but are able to slide tangentially relative to one another.



Contact of noncircular Beams in Abaqus/Standard and Abaqus/Explicit

2025 FD01

- Abaqus/Standard and Abaqus/Explicit uses an automatically generated mesh of contact faces, nodes, and edges close to the exterior of noncircular beam cross-sections.
- Some surface thickness is assigned automatically, which provides an accurate contact representation of the beam geometry with some rounding of edges.

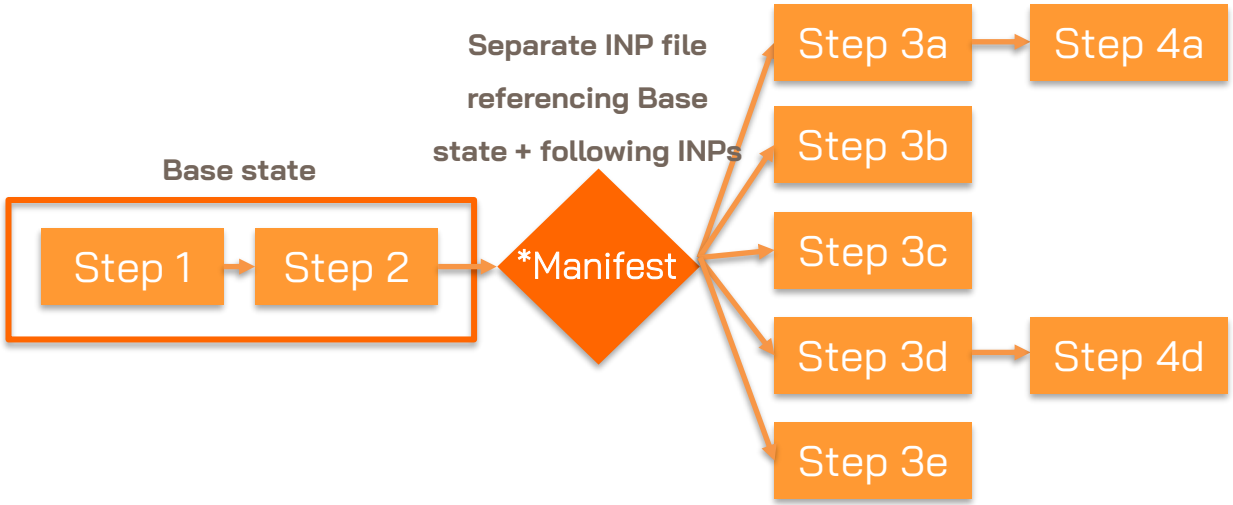


Contents

MANIFEST

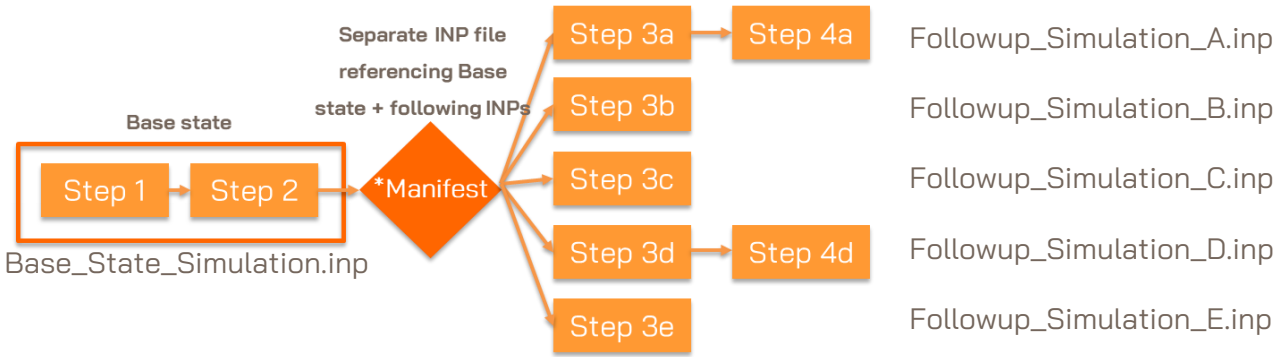
REVIEW AND WHEN TO USE

- Alternative to *LOAD CASE for non-linear problems
- Base state of simulation → load cases branching from it



Contents of Simulation_manifest.inp:

*MANIFEST, EVOLUTION TYPE=HISTORY, RESULTS=APPEND, BASE STATE=YES
Base_State_Simulation.inp
Followup_Simulation_A.inp
Followup_Simulation_B.inp
Followup_Simulation_C.inp
Followup_Simulation_D.inp
Followup_Simulation_E.inp



Run simulation with:

abaqus –job Simulation_manifest –cpus 8

MANIFEST PARAMETERS

*MANIFEST, EVOLUTION TYPE=HISTORY, RESULTS=APPEND, BASE STATE=YES, ZERO INCREMENT = YES, MODEL CHANGE

Required:

EVOLUTION TYPE=HISTORY. Each INP after first line defines steps forming a non-linear load case

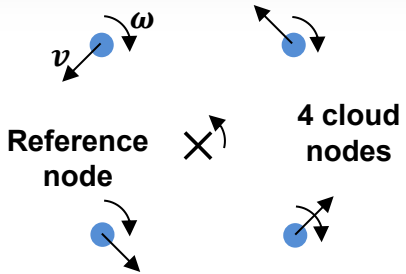
Optional:

RESULTS=APPEND/NEW	Create single ODB file./Create separate ODB files for each line.
BASE STATE=NO/YES	The initial state of the model is the base state./The state after line 1 is the base state.
ZERO INCREMENT=YES/NO	Write zero increment to ODB./Do not write zero increment to ODB.
MODEL CHANGE	Include if element removal or reactivation occurs during any nonlinear load case.

ENHANCED DISTRIBUTING COUPLING

MODIFICATION

- **Distributing coupling** is an "averaging constraint"
  - Translation of reference node is coupled to average translation of the cloud node system
  - Rotation of reference node is coupled to average rotation of the cloud node system



- **Measures how to influence behavior of distributing coupling**

*DISTRIBUTING, COUPLING= influence how translation of the reference node is coupled to cloud nodes

*DISTRIBUTING, ROTATIONAL COUPLING= influence how rotation of the reference node is coupled to cloud nodes

➔ Ability to couple reference node rotation to colinear structural cloud nodes in Abaqus/Explicit 2025 FD01 (Abaqus/Standard in 2020 FD01) .

ENHANCED DISTRIBUTING COUPLING MODIFICATION

- New option for Rotational Coupling for Abaqus/Explicit 2025 FD01

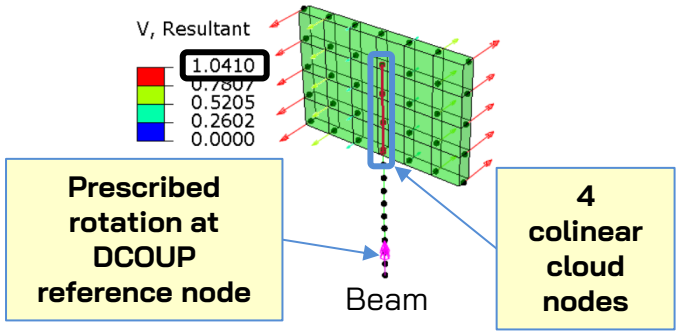
  *Distributing, Rotational Coupling=**Automatic** **2025 FD01**
  - reference node rotation respect cloud nodes rotations only if cloud nodes are (approximately or actually) colinear

  *Distributing, Rotational Coupling=Continuum
  - reference node rotations respect only translations of cloud nodes

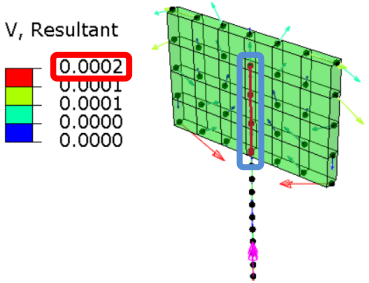
  *Distributing, Rotational Coupling=Structural
  - reference node rotations respect translations and rotations of cloud nodes

- Option Automatic default setting for Abaqus/Std & /Exp **2025 FD01**

Rotational Coupling=Automatic
(or Rotational Coupling=Structural)

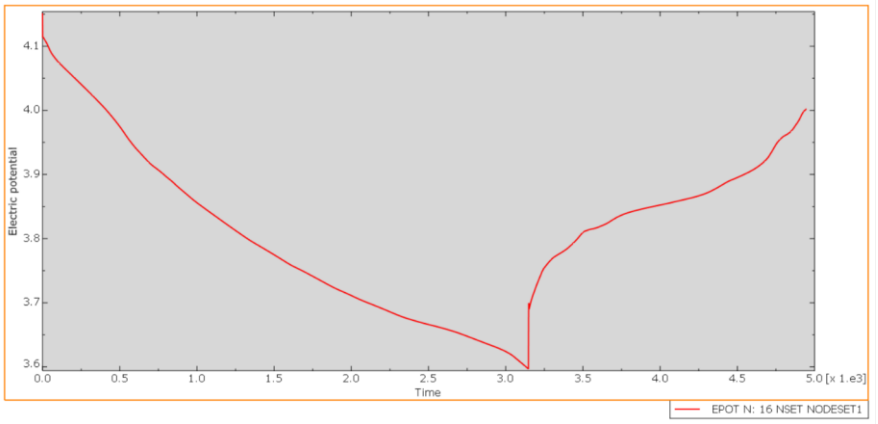


Rotational Coupling=Continuum



STEP CONTROL
REVIEW

- **Step Control**
  - Terminate the analysis when the solution variables reach or exceed a specified threshold
  - Control a simulation that has multiple steps in **Abaqus/Standard**
- **Example – a battery analysis / multiple steps**
  - Terminate a step at a specified voltage (or) current



*NSET, NSET = REACTION_FORCE

*STEP
*STATIC
1, 1, 1E-5, 1
...
***STEP CONTROL**, NAME = CONTROL1, ACTION = END STEP
Monitor1, ABSMAX, 3.6

*OUTPUT, HISTORY, **SENSOR**, NAME=Monitor1, FREQ=1
*NODE OUTPUT, NSET = REACTION_FORCE
RF

- **Control of reaching of given threshold value for more accurate results**
  - Refinement of time incrementation when reaching threshold value by DF REFINEMENT parameter

    DT REFINEMENT=YES (default - user-specified fixed time increment should be used)

    DT REFINEMENT=NO (to leave the time increment unaffected)

    DT REFINEMENT=AUTO (Aba/Std should refine the time increment automatically to reach the threshold as closely as possible)

- **Enhancement for DT REFINEMENT=AUTO**
  - The default tolerance value of relative "error" to determine if the sensor has reached the specified threshold is **1.0e-3**.
  - A new parameter **TOLERANCE** to have control over this value **2025 FD01**

*STEP CONTROL, ACTION= End Analysis, NAME= Example, DT REFINEMENT = AUTO, **TOLERANCE =1.0e-4**

**sensorName, Criteria, Threshold Value

Monitor1, MAX, 1

Step control vs. Extreme value

Comparison

Step control

```
*NSET, NSET = REACTION_FORCE

*STEP
*STATIC
1, 1, 1E-5, 1
...
*STEP CONTROL, NAME = CONTROL1, ACTION = END STEP
Monitor1, ABSMAX, 17
*OUTPUT, HISTORY, SENSOR, NAME=Monitor1, FREQ=1
*NODE OUTPUT, NSET = REACTION_FORCE
RF1
```

Extreme value

```
*NSET, NSET = REACTION_FORCE

*STEP
*DYNAMIC, EXPLICIT
, 0.1
...
*EXTREME VALUE, HALT=YES
*EXTREME NODE VALUE, NSET = REACTION_FORCE, ABS
RF1, 17
```

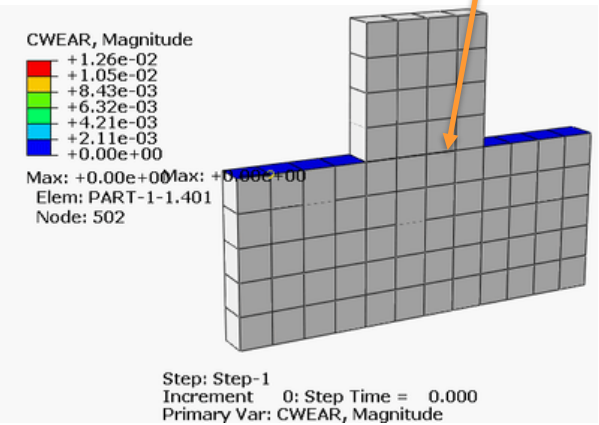
STEP CYCLING
IN ABAQUS/STANDARD

2025 GA

- The step cycling capability provides a simple way to repeat a step or a series of steps over a prescribed number of cycles.
- Examples: aging in batteries, brake wear analysis, fatigue
- Loop over the sequence of steps with significant performance improvements
- Provides mechanisms to end a step sequence prematurely based on certain user-specified criteria (*STEP CYCLING CONTROL)
- Benefits: Simpler model setup, better performance, ability to end step cycling based on the analysis solution.

```
*STEP CYCLING, START, NAME=cycname
number of cycles, frequency at which to write output, frequency at which to create a new output database
*STEP
Input defining the first step of the step cycling sequence
*END STEP
Repeat as necessary for additional steps within the same step cycling sequence
*STEP CYCLING, END
```

Wear accumulation



Contents

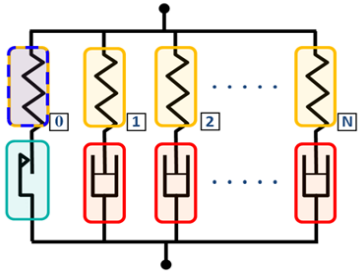
ENHANCEMENTS TO PRF REVIEW

- **Parallel Rheological Framework (PRF)**
  - allows modeling polymers and elastomeric materials that exhibit permanent set and nonlinear viscous behavior and undergo large deformations.

- **PRF material model**
  - multiple viscoelastic networks and, optionally, an elastic-plastic network in parallel
  - hyperelastic material model to specify the elastic response
  - can be combined with Mullins effect
  - can include nonlinear kinematic hardening in the elastic-plastic response

- **New user defined subroutine (VUHYPER)**
  - at network level Network ID information within user subroutine
  - Different hyperelastic models for different networks

PRF material model



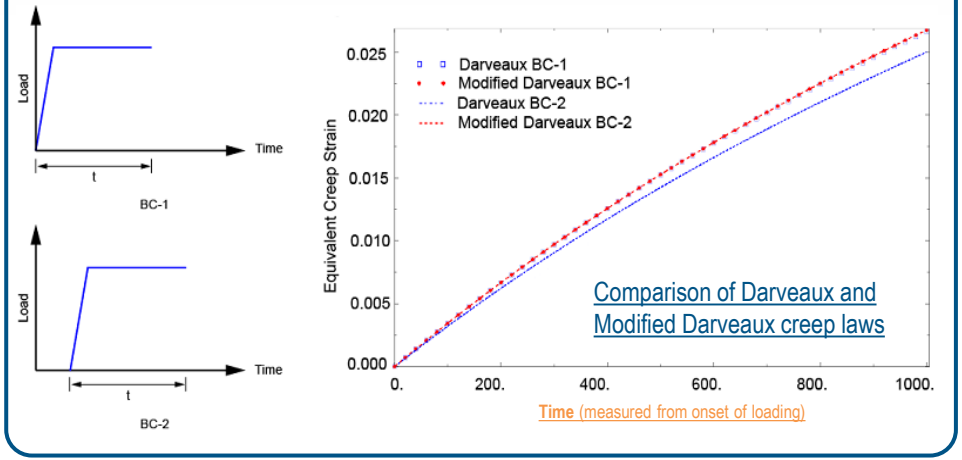

```
*MATERIAL
*HYPERELASTIC, USER
*PLASTIC
*VISCOELASTIC,NONLINEAR,NTEWORKID=1,LAW=...
*VISCOELASTIC,NONLINEAR,NTEWORKID=2,LAW=...
...
```

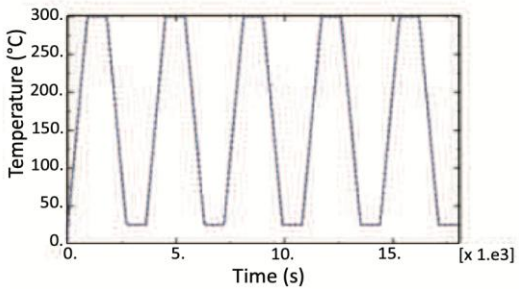
- **New Hencky hyperelasticity model in Aba/Std**
  - Possibility to model hyperelastic material response accurately

ENHANCEMENTS TO PRF

ENHANCEMENT

- **New creep laws**
  - Darveaux creep
    - primary and secondary creep, but introduces unphysical dependency on time
    - Useful for the calibration of thermoplastic materials such as Polyamide-12 (PA-12)
  - Modified Darveux creep
    - Eliminates explicit dependency on time, producing the same response of the Darveaux model under constant load and temperature
  - Anand creep
    - Suitable for cyclic creep states





PAPERBOARD MATERIAL MODEL

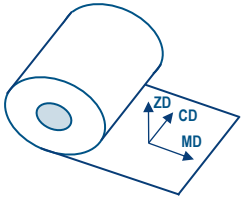
Aba/Std/Exp 2025 FD01

- **Paperboard material's fibers orientation (orthotropic) and local directions**
  1. The machine direction (MD) - most of the fibers
  2. The cross direction (CD) direction

  } in-plane principal directions

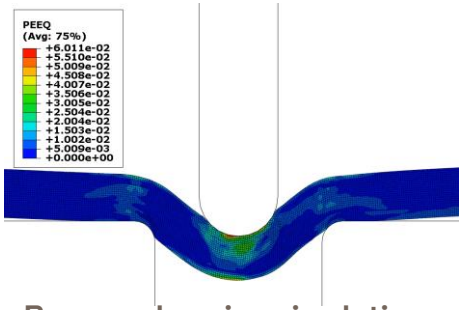
  3. The thickness direction (ZD) - almost no fibers

  } Out-of-plane principal directions

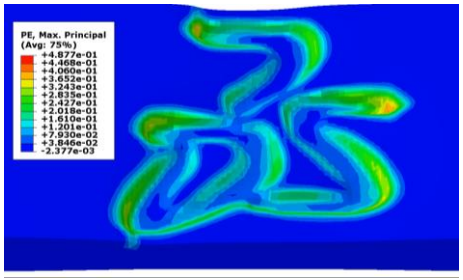


- **New paperboard plasticity model available in Aba/Std/Exp 2025 FD01**
  - In-plane response (MD, CD, and MD-CD shear):
    - Orthotropic linear elasticity with orthotropic plasticity and hardening
    - Allows modeling different yield in tension and compression for each direction and the in-plane shear.
  - Decoupled out-of-plane shear (MD-ZD and CD-ZD) with plasticity and hardening
  - Decoupled porous elastic response in the ZD direction with plasticity in the ZD compression
- **Supported with 3D continuum and shell elements**
- **Simulations of workflows containing paperboard material, such as paper creasing and embossing in the packaging industry**

Paper creasing simulation



Paper embossing simulation



- **Complete material definition:**

*ELASTIC, TYPE=ORTHOTROPIC/ENGINEERING CONSTANTS

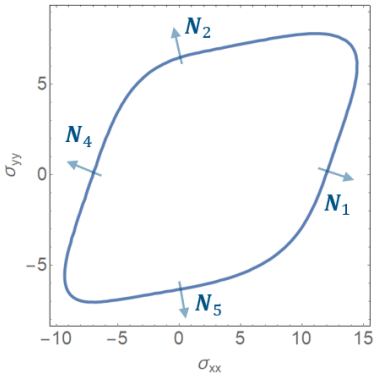
...

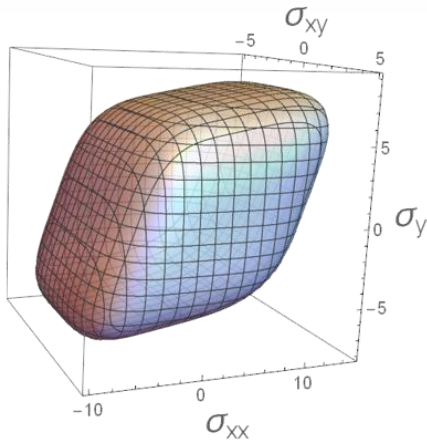
in-plane response

*PAPERBOARD PLASTICITY, EXPONENT=$2k$, DEPENDENCIES=
$v_{1p}, v_{2p}, v_{4p}, v_{5p}$,temp,fv1,...

*PAPERBOARD HARDENING,DEPENDENCIES=
$\sigma_1^0, A_1, B_1, C_1, \sigma_2^0, A_2, B_2, C_2,$
$\sigma_3^0, A_3, B_3, C_3, \sigma_4^0, A_4, B_4, C_4,$
$\sigma_5^0, A_5, B_5, C_5,$,temp,fv1,...

$$\bar{\sigma}^I = \bar{\sigma}_0^I + A_I \tanh\left(B_I \bar{\varepsilon}^{pl}\right) + C_I \bar{\varepsilon}^{pl}$$





Yeild surface by Xia

Out-of-plane response

*PAPERBOARD THICKNESS COMPRESSION ELASTIC,VOID RATIO=r_{orig},DEPENDENCIES=
μ,σ_z^t,temp,fv1,...

*PAPERBOARD THICKNESS COMPRESSION PLASTICITY,DEPENDENCIES=
$A_\sigma, B_\sigma, C_\sigma$, temp,fv1,...

$$\sigma_s = A_\sigma + B_\sigma \exp\left(-C_\sigma \varepsilon_{33}^{pl}\right)$$

Out-of-plane Shear response

*PAPERBOARD TRANSVERSE SHEAR PLASTICITY,DEPENDENCIES=
A_τ, B_τ, C_τ, temp,fv1,...

$$\tau_s = A_\tau + B_\tau \tanh\left(C_\tau \bar{\gamma}^{pl}\right)$$

PAPERBOARD MATERIAL MODEL

Aba/Std/Exp 2025 FD01

Example: Box compression

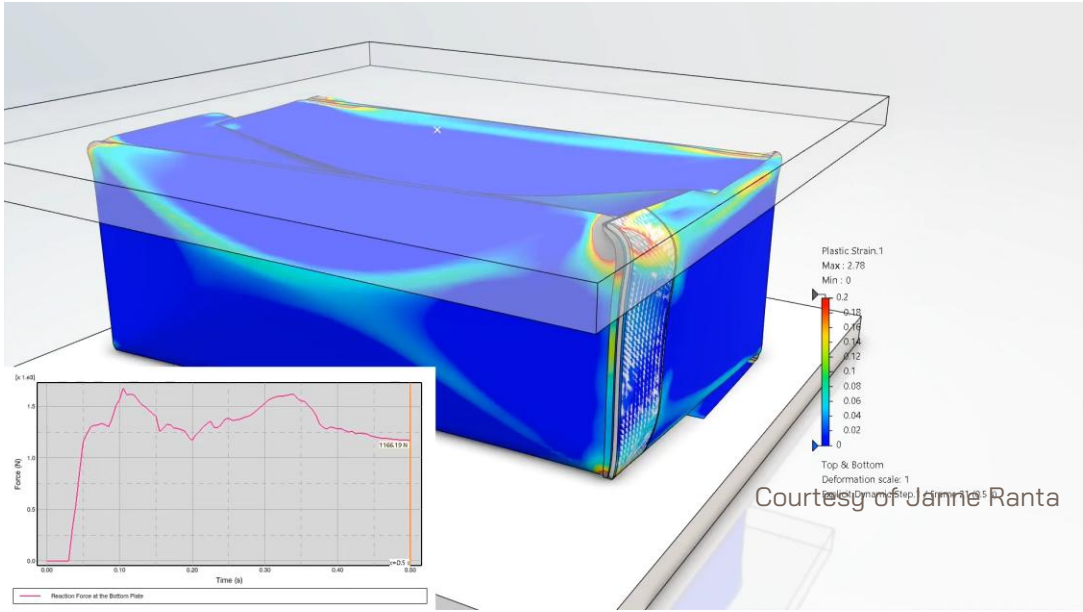
- **Complete material definition:**

*ELASTIC, TYPE=ORTHOTROPIC/ENGINEERING CONSTANTS
...

in-plane responce:
*PAPERBOARD PLASTICITY, EXPONENT=$2k$, DEPENDENCIES=
$v_{1p}, v_{2p}, v_{4p}, v_{5p}$,temp,fv1,...
*PAPERBOARD HARDENING,DEPENDENCIES=
$\sigma_1^0, A_1, B_1, C_1, \sigma_2^0, A_2, B_2, C_2,$
$\sigma_3^0, A_3, B_3, C_3, \sigma_4^0, A_4, B_4, C_4,$
$\sigma_5^0, A_5, B_5, C_5,$,temp,fv1,...

Out-of-plane responce:
*PAPERBOARD THICKNESS COMPRESSION ELASTIC,VOID RATIO=r_{orig},DE
μ,σ_z^t,temp,fv1,...
*PAPERBOARD THICKNESS COMPRESSION PLASTICITY,DEPENDENCIES=
$A_\sigma, B_\sigma, C_\sigma$, temp,fv1,...

Out-of-plane Shear responce:
*PAPERBOARD TRANSVERSE SHEAR PLASTICITY,DEPENDENCIES=
A_τ, B_τ, C_τ, temp,fv1,...

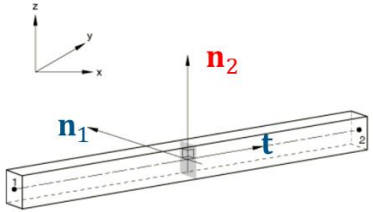


Courtesy of Janne Ranta

Beam Normal

Review

- **General beam cross-sectional axis** (local, right-handed)



t is the tangent to the element axis

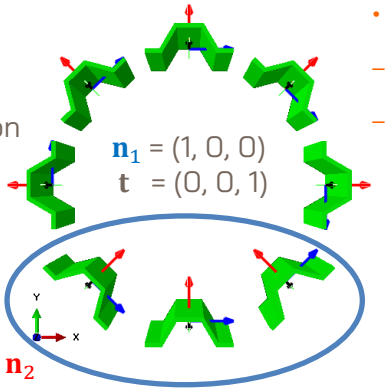
n_1 is the local 1 – direction of cross-section

n_2 is the local 2 – direction of cross-section

is the normal to the beam

- **Approximate Local n_1 – direction of cross-section**
  - For beams in a plane is always (0, 0, –1)
  - For beams in space must be defined:
    - directly in the beam section definition

      or
    - by specifying an additional node



Rule Based Reversed n_2

- **Local 2 – direction of cross-section**
  - For beams in space („rule based" approach) user can:
    - specify nodal normal by direction cosines of each node

      or
    - user-specified normal definition
    - Otherwise, the nodal normal will be calculated by Abaqus

- **Actual Local 1 – direction of cross-section**
  - calculated actual $n_1 = n_2 \times \mathbf{t}$.
  - **"Rule-based" approach**
    - If the angle between n_2 and $\mathbf{t} \times n_1$ is greater than 90°, the **user specified n_2 – direction is reversed**

potential discontinuities in the beam normal in different areas in the model

BEAM NORMAL ENHANCEMENT

- **New feature how to control behavior of beams in space nodal normals** **2025 FD01**

  *NORMAL, TYPE =ELEMENT, **BEAM NORMAL=RULE BASED (default)**

  *NORMAL, TYPE =ELEMENT, **BEAM NORMAL=FIXED**

  – enforce that the normal used by Abaqus is exactly as you defined it (and not reversed by Rule-based approach)

Input example:

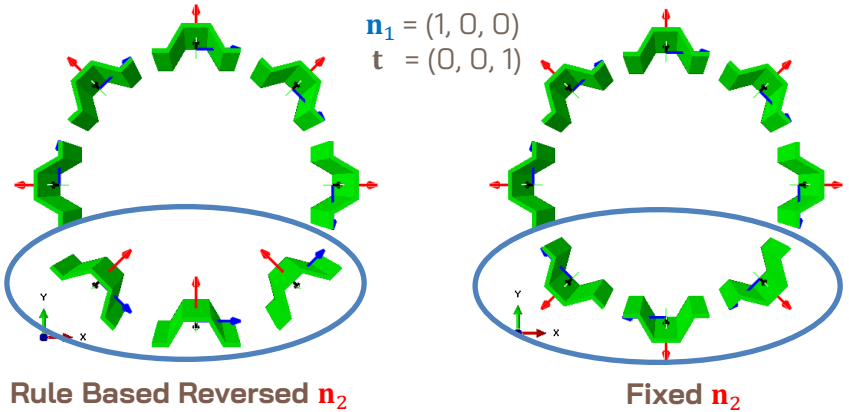
*NORMAL, TYPE =ELEMENT, BEAM NORMAL=**FIXED**

element 1, node 1, X1, Y1, Z1 (components of normal vector)

element 2,node 2, X2, Y2, Z2 (components of normal vector)

or

ELSET, NSET, X, Y, Z (components of normal vector)



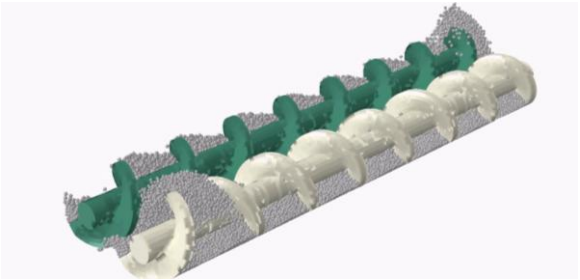
DEM Particle clustering

DEM overview

- **The Discrete Element Method (DEM) in Aba/Exp only**
  - Discrete particles / clusters collide with each other and with other surfaces
  - DEM particle represents a separate gravel, grain, tablet, shot peen, etc.
  - not applicable to situations in which individual particles undergo complex deformation
  - DEM can be used together with finite elements for modeling discrete particles interacting with deformable continua or other rigid bodies

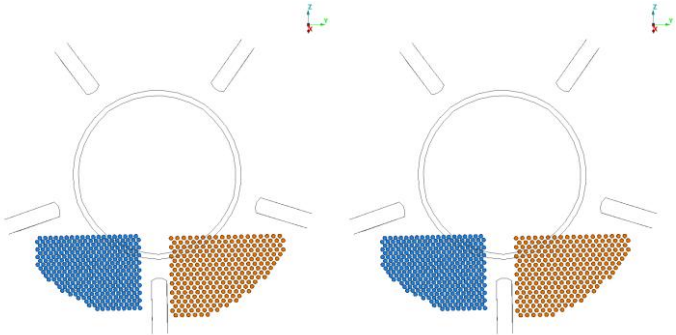
- **Applications**
  - pharmaceutical, chemical, food, ceramic, metallurgical, mining, ...
  - Particle packing, pouring or deposition under gravity (such as sandpiling)
  - vibration after deposition of particles, Compaction
  - Particle flow, under gravity only (as in the case of a hopper) and under gravity and other driving forces (such as for mixers and mills).

Auger mixer



Rotating drum mixer

Non-adhesive particles | Adhesive particles



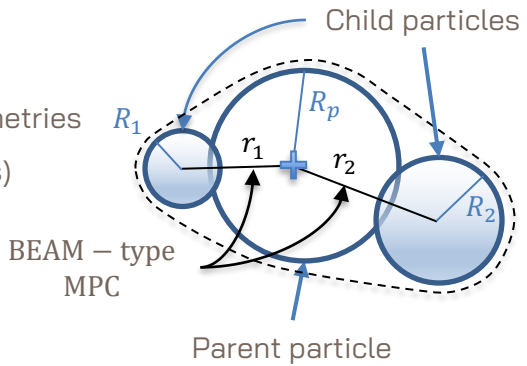
DEM PARTICLE CLUSTERING

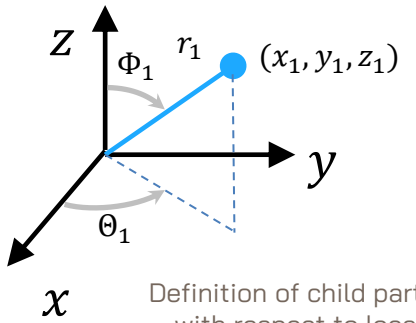
CLUSTERING

- **DEM particle-cluster approach**
  - Benefits:
    - More realistic representation of arbitrarily-shaped (non-spherical) particle geometries
    - Improves model predictions (Grain locking, Dilation effects under shearing loads)
  - Consists of a parent particle and one or more child particles
  - Parent particle = PD3D element
  - Child particle defined in the *DISCRETE SECTION by position and size
  - Particles in a cluster might overlap with each other
  - Held together either rigidly or via compliant connections

*DISCRETE SECTION, ELSET=DEM1
R_p ← Parent particle size
$\alpha_1, \beta_1, \Theta_1, \Phi_1$
$\alpha_2, \beta_2, \Theta_2, \Phi_2$ } ← 1 line per child particle
...
*CLUSTER MASS INERTIA TABLE $M_1, \boldsymbol{I}_1, R_{p1}$
$M_2, \boldsymbol{I}_2, R_{p2}$
... } ← Mass and inertia tensor of cluster as function of parent particle size, R_p

Cluster of particles

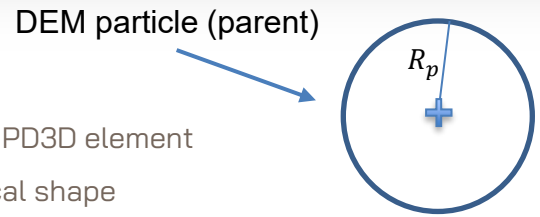




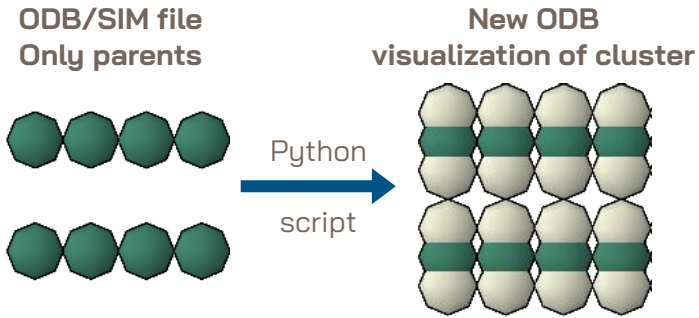
Child particle 1

$r_1 = \alpha_1 R_p$
$R_1 = \beta_1 R_p$
$x_1 = r_1 \sin(\Phi_1) \cos(\Theta_1)$
$y_1 = r_1 \sin(\Phi_1) \sin(\Theta_1)$
$z_1 = r_1 \cos(\Phi_1)$

Definition of child particle 1 size and location with respect to local system centered at parent particle center



- **DEM particle**
  - single-node PD3D element
  - rigid spherical shape
  - radius and density (*DISCRETE SECTION)
    → mass and rotational inertia are calculated automatically
  - displacement and rotational DOF
- **Visualization**
  - ODB/SIM file only contain parent particle and cluster section information
  - A python script is available to generate new ODB for visualization of all cluster particles
- **Output**
  - nodal output includes all output variables
  - No element output is available for PD3D elements
  - Resultant of all contact forces only for parent particle
- **Interactions**
  - general contact as element-based surfaces
  - Possible to adjust surface behavior of contact between particles
    - Normal and tangential contact stiffness
    - Hertz contact normal stiffness
    - Adhesive Normal Contact (Johnson-Kendall-Roberts)



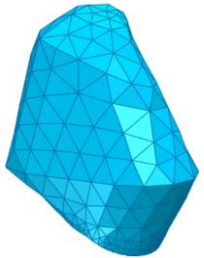
DEM PARTICLE CLUSTERING ENHANCEMENT

Aba/Exp 2024 FD01 & 2025 FD01

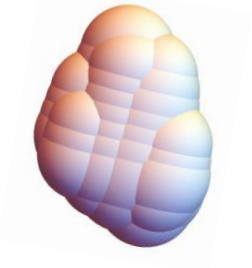
- **Abaqus Python is now based on Python 3.10 instead of Python 2.7**
- **Script available for generation of particle clusters**
  - Semi-automatic procedure for creating clusters
  - Create DEM cluster definition for complex shapes
  - User control on cluster approximation of the original shape by including more child particles
  - The mass and inertial property table for the cluster is automatically created by the script

- **Generation workflow**
  - FE discretization of the shape.
  - DC with the provided uinter.for subroutine
  - Run the python script with .log file from DC
  - Include the section data lines generated by the script in DEM model.
  - Run your DEM model and visualize the results

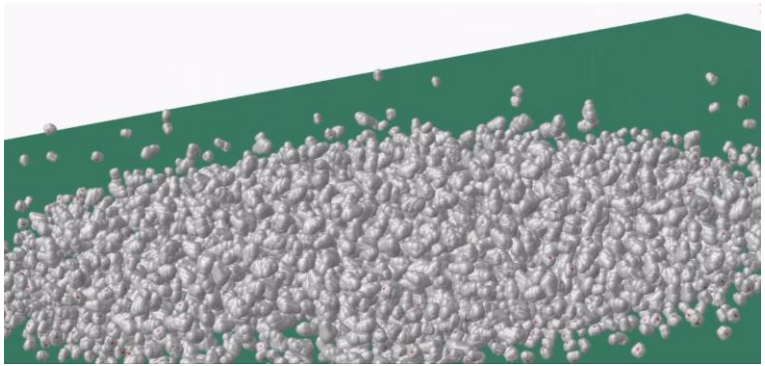
FE mesh



Cluster approximation



Falling rocks simulation



ALTERNATIVE CONVERGENCE MEASURES IN ABAQUS/STANDARD

- **CONVERGENCE CHECK=STRICT (Default)**
  - Ratio of largest residual flux to the time averaged flux is less than a tolerance
    and
  - Ratio of largest correction to the largest increment is less than a tolerance.

traditional convergence checks in Abaqus/Standard

work well for structural simulations, but might be too restrictive for some multiphysics problems and possibly leading to a large number of iterations

- **CONVERGENCE CHECK= MODERATE**
  - Ratio of largest residual flux to the largest flux is less than a tolerance, and
  - Ratio of largest correction to the largest increment is less than a tolerance.
- **CONVERGENCE CHECK= RELAXED**
  - The ratio of largest correction to largest increment is less than a tolerance
- **CONVERGENCE CHECK= CONCEPT DESIGN**
  - Ratio of largest residual flux to the time averaged flux is less than a tolerance
    or
  - Ratio of largest correction to the largest increment is less than a tolerance.

new set of convergence checks

2024 FD03

Benefits:

for simulations that involve strong multiphysics coupling among a number of different fields. Might reduce the solution time with minimal change in solution accuracy.

Important:

requires some experience and testing to ensure an acceptable level of accuracy

ALTERNATIVE CONVERGENCE MEASURES IN ABAQUS/STANDARD

- **Performance DATA*) with alternative convergence measures**

Features	Default (Strict)	Relaxed	Modified	Concept Design
XFEM	1280	794	1264	**674**
Fatigue	2288	2300	**2287**	2288
Geomechanics Fracture	**2777**	2919	2818	2808
Hydro-Fracture	609	**501**	Not Run	Not Run
Battery1	697	**367**	Not Run	Not Run
Battery2	1417	**958**	Not Run	Not Run
Thermal-stress	80	**56**	Not Run	Not Run
Static – Rubber gasket	253	**242**	Not Run	Not Run

*) Number of increments to complete the analysis

- Different convergence checks can be used for different fields in each step of the analysis.
- Convergence checks can be changed from one step to another
- CONCEPT DESIGN, can be used for running a range of models to evaluate the best initial design (design of experiments).
- Solution speedup by up to 25% for battery multiphysics simulations, using the “RELAXED” criteria

Contents

GENERATE NODE SET FROM SURFACE

2024 FD01

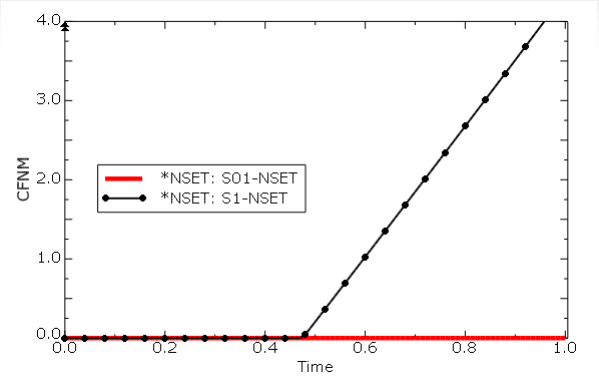
- Specify the name of an existing surface
- Nodes of the surface will be assigned to the specified node set
- Benefits: This surface-based node set feature makes it easier to create a node set that includes all the nodes on a predefined surface.

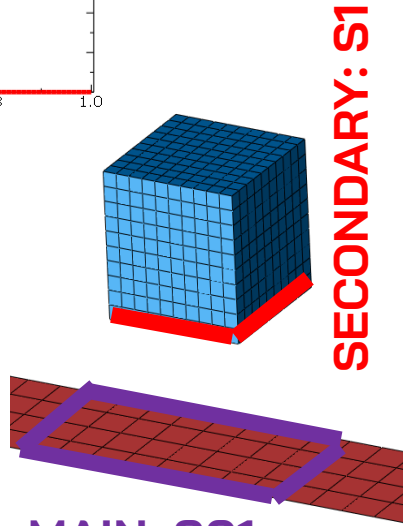
```
*SURFACE, NAME=surf_top
*NSET, NSET=A15, SURFACE=surf_top
```

OUTPUT

HISTORY OUTPUT REQUESTS

***SURFACE INTERACTION**, NAME=INTPROP-1
***FRICTION**, SLIP TOLERANCE=0.005
0.1
***SURFACE BEHAVIOR**, PENALTY=NONLINEAR
***CONTACT**
***CONTACT INCLUSIONS**
S1, S01
***CONTACT PROPERTY ASSIGNMENT**
, , INTPROP-1
***CONTACT FORMULATION**, TYPE=MAIN SECONDARY ROLES
S1, S01, SECONDARY
...
***OUTPUT**, HISTORY
***CONTACT OUTPUT**, NSET=S1-NSET
CFN, CAREA





- Contact output je zapisován pouze z uzlů (NSET) náležící secondary surface
- Je vhodné manuálně definovat co je Main/Secondary surface

ANIMATION FILE FORMATS IN ABAQUS/CAE

2025 FD01

- You can now save and load animations in MP4 and animated GIF formats in Abaqus/CAE.
- Benefits: New file formats offer improved compatibility with modern media players and editing software, as well as HTML5 and common applications for producing presentations. The file size can be significantly smaller, with no appreciable loss of quality.

IMPORTING ALL ELEMENT SETS FROM PREVIOUS ANALYSIS

2025 FD03

- You can now easily import all element sets in the previous analysis that have a section definition that is supported for import.
- The new, default input method imports elements once from all importable sections defined in a previous analysis. It also supports repositioning of the imported elements and importing from multiple previous analyses.
- Benefits: You can now import all elements in the analysis model without explicitly listing the element sets to be imported.

```
*IMPORT
list of element sets that are to be imported and relocated
positioning data
```

```
*IMPORT
blank line
positioning data
```

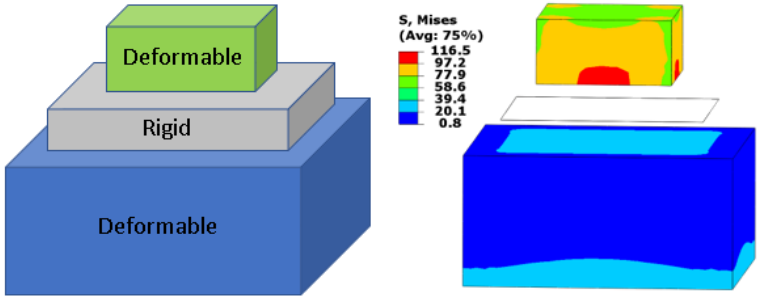
SURFACE ELEMENT THICKNESS

2025 FD01

- You can now define a constant section thickness as part of the surface section definition.
- By default, user-defined surface elements have zero element thickness.
- Any specified section thickness only affects contact calculations.

```
*SURFACE SECTION, ELSET=name
thickness
```

- Element calculations (including rebar layers for defining reinforcements or mass and rotary inertia properties, for example) are computed as if the section thickness is zero.
- Benefits: Individual surface elements with finite thickness can represent a volume for contact calculations, similar to the effects of shell and membrane element thickness, enabling you to better model contact with surface elements.



Environment File

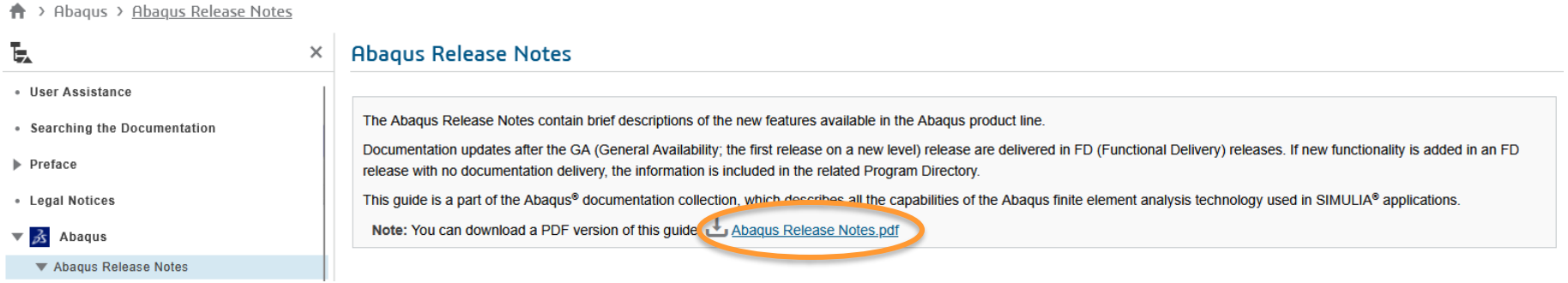
Customize Warnings Level

2025 FD01

- The Abaqus analysis input file processor now suppresses common warnings for unsupported output requests by default. However, you can use the new environment variable **output_warnings_level = ALL/OFF/DEFAULT** to specify if common warnings are reported.
- Many warnings issued by the Abaqus analysis input file processor for invalid output requests are informational only and warrant no action by the user.
- For large models, the number of these warnings can be very high and can make it difficult to evaluate the preprocessed model.
- Benefits: Reducing the number of warnings makes evaluation of the preprocessed model more convenient.

2025 GA

- Since version 2025 GA a PDF manual is available for download
- One manual chapter = 1 PDF file
- help.3ds.com



Applus IDIADA

THANK YOU

FOR YOUR KIND ATTENTION

www.applusidiada.com

Applus IDIADA

Headquarters and Main Technical Centre

L'Albornar – PO Box 20
E-43710 Santa Oliva (Tarragona) Spain
T +34 977 166 000 F +34 977 166 007
e-mail: idiada@idiada.com

www.idiada.com

Applus IDIADA Belgium
T +32 2 757 27 07 (Brussels)
e-mail: idiada_belgium@idiada.com

Applus IDIADA Brazil
T +55 11 4330 9880 (São Paulo)
T +55 31 3591 6832 (Belo Horizonte)
e-mail: idiada_brasil@idiada.com

Applus IDIADA China
T +86 21 6210 0894 (Shanghai)
T +86 21 6210 0894 (Beijing)
T +86 431 8190 9680 (Changchun)
T +86 23 6756 8060 (Chongqing)
T +86 21 6210 0894 (Cixi)
T +86 20 2282 9202 (Guangzhou)
T +86 21 6210 0894 (Ningbo)
T +86 532 66019017 (Qingdao)
T +86 21 6210 0894 (Tianjin)
T +86 21 6210 0894 (Wuhu)
T +86 535 8933658 (Zhaoyuan)
e-mail: idiada_china@idiada.com

Applus IDIADA Czech Republic
T +420 493 654 811 (Hradec Králové)
T +420 778 430 095 (Brno)
T +420 482 424 243 (Liberec)
T +420 326 736 860 (Mladá Boleslav)
e-mail: info@idiada.cz

Applus IDIADA France
T +33 (0) 181 891 943 (Paris)
T +33 (0) 130 370 836 (Paris)
T +33 (0) 141 146 085 (Lyon)
e-mail: idiada_france@idiada.com

Applus IDIADA Germany
T +49 (0) 84188538-30 (Ingolstadt)
T +49 (0) 89309056-0 (Munich)
T +49 (0) 84188538-30 (Stuttgart)
T +49 (0) 5374920606-0 (Wolfsburg)
e-mail: idiada_germany@idiada.com

Applus IDIADA India
T +91 44 2275 2202 (Chennai)
T +91 124 4028 888 (New Delhi)
T +91 20 6605 6800 (Pune)
e-mail: idiada_india@idiada.com

Applus IDIADA Italy
T +39 051 0923530 (Bologna)
T +39 005 10923500 (Erbusco)
T +39 011 2640320 (Turin)
e-mail: idiada_italia@idiada.com

Applus IDIADA Japan
T +81 (0) 42 512 8982 (Tokyo)
T +81 (0) 52 588 5329 (Nagoya)
e-mail: idiada_japan@idiada.com

Applus IDIADA Malaysia
T ++60327281027 (Kuala Lumpur)
T +601 2410 7686 (Penang)
e-mail: idiada_malaysia@idiada.com

Applus IDIADA Mexico
T +52 (222) 644 1374 (Puebla)
e-mail: idiada_mexico@idiada.com

Applus IDIADA Morocco
e-mail: idiada_morocco@idiada.com

Applus IDIADA Poland
T +48 61 6226 905 (Poznan)
e-mail: idiada_polska@idiada.com

Applus IDIADA Slovakia
T +420 778 430 098 (Košice)
e-mail: idiada_slovakia@idiada.com

Applus IDIADA South Korea
T +82 31 478 1821 (Seoul)
e-mail: idiada_korea@idiada.co.kr

Applus IDIADA Spain
T +34 977 166 000 (Santa Oliva)
T +34 928 587 447 (Las Palmas)
T +34 915 095 795 (Madrid)
T +34 950 473 256 (Mojácar)
T +34 868 912 179 (Murcia)
T +34 948 292 921 (Pamplona)
T +34 955 117 111 (Sevilla)
T +34 986 900 300 (Vigo)
e-mail: idiada@idiada.com

Applus IDIADA Sweden
T +46 (0) 31 320 1844 (Gothenburg)
T +46 731 478 202 (Stockholm)
e-mail: idiada_sweden@idiada.com

Applus IDIADA Thailand
T +66 86 7917 071 (Bangkok)
e-mail: idiada_thailand@idiada.com

Applus IDIADA Turkey
T +90 216 250 6050 (Istanbul)
e-mail: idiada_turkey@idiada.com

Applus IDIADA UK
T +44 1223 441 434 (Cambridge)
T +44 2476 328 083 (Nuneaton)
T +44 1926 623 132 (Warwick)
e-mail: idiada_uk@idiada.com

Applus IDIADA UAE
T +971 4 2441313 (Dubai)
e-mail: idiada_uae@idiada.com

Applus IDIADA USA
T +1 248 978 0111 (Detroit)
T +1 760 246 1672 (Los Angeles)
e-mail: idiada_USA@idiada.com

Applus IDIADA Vietnam
T +84 97 724 19 86 (Hanoi)
e-mail: idiada_vietnam@idiada.com

Applus IDIADA

top EMPLOYER España Spain 2025 FOR A BETTER WORLD OF WORK

top EMPLOYER Brasil Brazil 2025 FOR A BETTER WORLD OF WORK

top EMPLOYER India 2025 FOR A BETTER WORLD OF WORK

top EMPLOYER United Kingdom 2025 FOR A BETTER WORLD OF WORK