

## What is new in ABAQUS 6.5

---

Version 6.5 is a major release of ABAQUS. There are significant new features and enhancements in all ABAQUS products, including the following:

- The Model Tree is a convenient new tool for navigating and managing models. The Model Tree appears in the ABAQUS/CAE main window, providing a visual description of the model object hierarchy. The Model Tree also provides shortcuts to the main menu bar, module toolboxes, and the various managers.
- ABAQUS/Explicit analyses using domain-level parallelization require significantly less memory per domain as the number of domains increases in Version 6.5 compared to Version 6.4. The memory required for single-domain analyses has also been reduced for Version 6.5.
- ABAQUS/Explicit offers a general capability for modeling progressive damage and failure in the Mises, Johnson-Cook, Hill, and Drucker-Prager plasticity models. Damage initiation criteria are available for ductile or shear fracture of metals and necking instability of sheet metal as well as damage initiation in cohesive elements with traction-separation response
- A family of cohesive elements is supported in ABAQUS/Standard, ABAQUS/Explicit, and ABAQUS/CAE for modeling deformation and damage in finite-thickness adhesive layers between bonded parts. Cohesive elements are typically connected to underlying elements with surface-based tie constraints, so the mesh used for the cohesive layer can be independent of the mesh used for the bonded components.
- Numerous enhancements have been implemented to improve contact performance and robustness in ABAQUS/Standard and ABAQUS/Explicit.
  - The accuracy of ABAQUS/Standard contact stress results improved with the addition of a true surface-to-surface formulation for small-sliding contact to mitigate contact stress oscillations at interfaces, even for mismatched meshes. The new formulation also accounts for surface thicknesses and offsets.
  - Noise in contact and element stresses near the contact interface is reduced for most second-order element types in ABAQUS/Standard by specifying a softened pressure-overclosure relationship or using an augmented Lagrangian contact enforcement method.
  - Improved stabilization methods can be specified for ABAQUS/Standard multi-body contact problems where rigid body modes exist initially because contact is not present in the first solution increment. You can apply viscous stabilization for both normal and tangential motions to selected contact pairs, including a damping coefficient based on the stiffness of the underlying material.
- The ability to define and analyze cracks in ABAQUS/CAE increases the coverage of ABAQUS fracture mechanics functionality.
  - ABAQUS/CAE includes support for fracture mechanics analysis using an embedded seam with duplicate overlapping nodes. The crack definition can be used to perform fracture calculations using contour integrals.
  - The position of the midside nodes of second-order elements can be adjusted using the ABAQUS/CAE Edit Mesh toolset to include a singularity at the crack tip for fracture mechanics analysis.

- In addition to improving boundary and surface meshing capabilities, ABAQUS/CAE Version 6.5 includes a completely new tetrahedral meshing algorithm.
  - Meshing a part in the Mesh module, as opposed to meshing a part instance from the assembly, allows the mesh to appear on each instance of the part although the part is meshed only once.
  - ABAQUS/CAE considers geometric curvature along edges to simplify the task of automatically seeding models that have holes of various sizes, blends, and other regions of high curvature with the appropriate target element size for meshing.
  - ABAQUS/CAE Version 6.5 includes a completely new tetrahedral meshing algorithm providing fast, reliable, automated meshing of very complex geometries. ABAQUS/CAE users have the option of selecting either the new (default) algorithm or the Version 6.4 tetrahedral meshing algorithm.
- View cuts enable visualization of the interior or selected portions of the model or results by creating planar, cylindrical, or spherical sections through the model. View cuts can also be defined along a constant contour variable value or isosurface.

ABAQUS Version 6.5 includes the following additional infrastructure enhancements:

- Version 6.5 is released on CD-ROM for the following combinations of supported operating systems and processors:
  - Linux/x86-32
  - Windows/x86-32
  - HP-UX/Itanium
  - Linux/Itanium
  - AIX/POWER

The full range of ABAQUS products is supported on Windows/x86-32 platforms. All ABAQUS products except fe-Safe Version 5 are supported on Linux/x86-32 platforms. The analysis products, ABAQUS/Standard and ABAQUS/Explicit as well as ABAQUS/Aqua and ABAQUS/Design, are supported on the remaining platforms listed above. For current details on supported ABAQUS products and platforms, see **Support & Services**→**Technical Support**→**Systems Information** on the ABAQUS Home Page.

- On Windows platforms ABAQUS/CAE supports native printing, which automatically lists available printers and selects the current default printer. Standard Windows print settings and dialog controls are used to choose page formatting and printer-specific options.
- 3Dconnexion motion controllers are supported in conjunction with the mouse to manipulate the model view with one hand on the motion controller while using the mouse to select from the model with the other hand. ABAQUS/CAE provides a set of “Application Functions” available through the standard 3Dconnexion user interface providing shortcuts to view manipulation and display tools.
- ABAQUS Version 6.5 uses FLEXlm Version 9.2 from Macrovision Corporation (upgraded from FLEXlm Version 8.2 for ABAQUS Version 6.4) for license management. For more information, see. [www.abaqus.com](http://www.abaqus.com)