

# Partial Differential Equation Toolbox™ Release Notes



# MATLAB®



## How to Contact MathWorks



Latest news: [www.mathworks.com](http://www.mathworks.com)  
Sales and services: [www.mathworks.com/sales\\_and\\_services](http://www.mathworks.com/sales_and_services)  
User community: [www.mathworks.com/matlabcentral](http://www.mathworks.com/matlabcentral)  
Technical support: [www.mathworks.com/support/contact\\_us](http://www.mathworks.com/support/contact_us)



Phone: 508-647-7000



The MathWorks, Inc.  
1 Apple Hill Drive  
Natick, MA 01760-2098

### *Partial Differential Equation Toolbox™ Release Notes*

© COPYRIGHT 2012–2022 by The MathWorks, Inc.

The software described in this document is furnished under a license agreement. The software may be used or copied only under the terms of the license agreement. No part of this manual may be photocopied or reproduced in any form without prior written consent from The MathWorks, Inc.

FEDERAL ACQUISITION: This provision applies to all acquisitions of the Program and Documentation by, for, or through the federal government of the United States. By accepting delivery of the Program or Documentation, the government hereby agrees that this software or documentation qualifies as commercial computer software or commercial computer software documentation as such terms are used or defined in FAR 12.212, DFARS Part 227.72, and DFARS 252.227-7014. Accordingly, the terms and conditions of this Agreement and only those rights specified in this Agreement, shall pertain to and govern the use, modification, reproduction, release, performance, display, and disclosure of the Program and Documentation by the federal government (or other entity acquiring for or through the federal government) and shall supersede any conflicting contractual terms or conditions. If this License fails to meet the government's needs or is inconsistent in any respect with federal procurement law, the government agrees to return the Program and Documentation, unused, to The MathWorks, Inc.

### **Trademarks**

MATLAB and Simulink are registered trademarks of The MathWorks, Inc. See [www.mathworks.com/trademarks](http://www.mathworks.com/trademarks) for a list of additional trademarks. Other product or brand names may be trademarks or registered trademarks of their respective holders.

### **Patents**

MathWorks products are protected by one or more U.S. patents. Please see [www.mathworks.com/patents](http://www.mathworks.com/patents) for more information.

## R2022b

<b>Interactive Solution Plots: Create and explore visualizations of PDE results</b> .....	1-2
<b>DC Conduction Analysis: Find stationary current distribution in conductors due to applied voltage</b> .....	1-2
<b>Magnetostatic Analysis: Model permanent magnets</b> .....	1-2
<b>Geometries: Import 3-D geometries from STEP files</b> .....	1-2

## R2022a

<b>Electromagnetic Analysis: Solve 2-D and 3-D wave scattering and transmission problems</b> .....	2-2
<b>Reduced-Order Modeling in Thermal Analysis: Approximate dynamic characteristics of a thermal model for faster execution</b> .....	2-2
<b>Hysteretic Damping: Model energy loss due to internal friction</b> .....	2-2
<b>Functionality being removed or changed</b> .....	2-2
pdecont will be removed .....	2-2
pdesurf will be removed .....	2-3

## R2021b

<b>Electromagnetic Analysis: Solve 3-D electrostatic and magnetostatic problems</b> .....	3-2
<b>Local Mesh Refinement: Set target mesh sizes around specified vertices, edges, and faces</b> .....	3-2
<b>Structural and Thermal Analysis: Extract sparse linear models for use with Control System Toolbox</b> .....	3-2

<b>PDEVisualization Properties: Display or hide axes and control background color</b> .....	3-2
---	-----

## R2021a

<b>Electromagnetic Analysis: Solve 2-D electrostatic and magnetostatic problems</b> .....	4-2
<b>3-D Geometries: Create a multidomain geometry by inserting one geometry into another</b> .....	4-2
<b>Geometry Extrusion: Extrude faces of a 3-D geometry to a specified height</b> .....	4-2
<b>Geometry Queries: Find faces, edges, and cells based on their proximity to other geometric entities</b> .....	4-3
<b>Geometry Import from STL Files: Import a geometry without creating a model</b> .....	4-3
<b>PDEVisualization object: Create and plot an object containing nodal data, mesh, and visualization options as its properties</b> .....	4-3
<b>Face Addition and Geometry Transformation for Analytic Geometries: addFace, rotate, scale, and translate now work with AnalyticGeometry objects</b> .....	4-3
<b>PDE Conversion to Divergence Form: Convert PDEs into the form required by Partial Differential Equation Toolbox using Symbolic Math Toolbox functions</b> .....	4-3

## R2020b

<b>3-D Geometry Creation: Extrude a 2-D geometry into a 3-D geometry</b> ...	5-2
<b>Finite Element Matrices for Custom Workflows: Assemble FE matrices for time-dependent and nonlinear models</b> .....	5-2

## R2020a

<b>Axisymmetric Analysis: Speed up simulations by simplifying 3-D solids of revolution by analyzing only the 2-D axisymmetric sections</b> .....	6-2
--	-----

<b>Multidomain Geometry: Split cells and fill voids to create multiple domains with different properties</b> .....	<b>6-2</b>
<b>Geometry Transformation: Manipulate geometries to preferred orientation and size by rotation, scaling, and translation</b> .....	<b>6-2</b>
<b>Modal Damping: Include damping in modal transient and frequency response simulations</b> .....	<b>6-2</b>
<b>pdegplot, pdemesh, pdeplot, and pdeplot3D Functions: Improved performance for plots with many text labels</b> .....	<b>6-3</b>

## **R2019b**

<b>Reduced Order Modeling: Approximate dynamic characteristics of a structural model for faster execution</b> .....	<b>7-2</b>
<b>Simscape Multibody Models: Generate dynamic substructure for flexible multibody simulations</b> .....	<b>7-2</b>
<b>Frequency Response Analysis: Determine dynamic response of a structure in the frequency domain</b> .....	<b>7-2</b>
<b>Structural Analysis: Set concentrated boundary conditions at arbitrary locations on geometry surfaces</b> .....	<b>7-3</b>
<b>Eigenvalue Solver Options: Specify parameters for Lanczos algorithm</b> .....	<b>7-3</b>

## **R2019a**

<b>Structural Analysis: Set boundary constraints and loads on edges and vertices</b> .....	<b>8-2</b>
<b>Structural Analysis: Compute transient response by using the modal superposition method</b> .....	<b>8-2</b>
<b>Finite Element Matrices: Assemble global finite element matrices for thermal and structural models</b> .....	<b>8-2</b>

## R2018b

<b>Thermal Load: Evaluate thermal stress and thermal expansion in static structural analysis</b> .....	<b>9-2</b>
--	------------

## R2018a

<b>Structural Analysis: Solve linear transient dynamic problems using direct time integration</b> .....	<b>10-2</b>
<b>Modal Analysis: Find natural frequencies and mode shapes</b> .....	<b>10-2</b>
<b>Mesh Analysis: Assess element quality, measure element area or volume, and find nodes and elements</b> .....	<b>10-3</b>
<b>Geometry from Mesh: Create multidomain geometry from nodes and elements</b> .....	<b>10-3</b>
<b>Functionality Being Removed or Changed</b> .....	<b>10-3</b>

## R2017b

<b>Structural Analysis: Solve static linear elasticity problems</b> .....	<b>11-2</b>
<b>Planar STL Geometry: Import and mesh planar STL geometries</b> .....	<b>11-2</b>
<b>Meshing: Improved mesh generation</b> .....	<b>11-2</b>
<b>Functionality Being Removed or Changed</b> .....	<b>11-3</b>

## R2017a

<b>Thermal Analysis: Solve conduction-dominant heat transfer problems with convection and radiation occurring at boundaries</b> .....	<b>12-2</b>
<b>3-D Multidomain Geometry: Create geometries consisting of stacked or nested cuboids, spheres, or cylinders</b> .....	<b>12-2</b>
<b>Functionality Being Removed or Changed</b> .....	<b>12-3</b>

<b>Flux of PDE Solution: Evaluate tensor product of c-coefficient and gradient of PDE solution for 2-D and 3-D problems</b> .....	13-2
<b>Boundary Conditions: Set an individual boundary condition for each equation in a PDE system, query boundary condition assignments</b> .....	13-2
<b>Nodal Initial Conditions: Set nodal initial conditions using the result of a previous simulation</b> .....	13-2
<b>Geometric Initial Conditions: Set initial conditions on vertices, edges, faces, and cells</b> .....	13-3
<b>Mesh and Solution Plots: Set transparency, display node and element labels</b> .....	13-3
<b>3-D Geometry Import from STL Files: Improved quality of the resulting geometry</b> .....	13-3
<b>Functionality Being Removed or Changed</b> .....	13-3

<b>PDE Solvers: Use solvepde and solvepdeeig functions to solve PDEs and PDE eigenvalue problems</b> .....	14-2
<b>PDE Coefficients: Specify equation coefficients as a property of PDEModel</b> .....	14-2
<b>Initial Conditions: Set initial conditions or initial guess as a property of PDEModel</b> .....	14-2
<b>Quadratic Elements for 2-D Mesh: Generate 2-D mesh using quadratic triangular elements</b> .....	14-2
<b>Gradient of PDE Solution: Evaluate solution gradient at arbitrary 2-D or 3-D points</b> .....	14-2
<b>Finite Element Matrices: Use assembleFEMatrices to assemble finite element matrices</b> .....	14-2
<b>PDE Results for Plotting and Postprocessing: New result objects depend on the type of PDE</b> .....	14-3
<b>Functionality Being Removed or Changed</b> .....	14-3

## R2015b

<b>3-D geometry creation from a finite element mesh using the geometryFromMesh function</b> .....	<b>15-2</b>
<b>Data structure to represent solutions using the createPDEResults function</b> .....	<b>15-2</b>
<b>Functionality Being Removed or Changed</b> .....	<b>15-2</b>

## R2015a

<b>3-D finite element analysis</b> .....	<b>16-2</b>
<b>Equation coefficients and boundary conditions for 3-D problems</b> .....	<b>16-2</b>
<b>Elliptic, parabolic, hyperbolic, nonlinear, eigenvalue solvers for 3-D problems</b> .....	<b>16-2</b>
<b>3-D geometry import from STL files</b> .....	<b>16-2</b>
<b>3-D unstructured meshing using tetrahedra</b> .....	<b>16-2</b>
<b>Plot function to inspect 3-D solutions</b> .....	<b>16-2</b>
<b>Featured examples with 3-D geometry</b> .....	<b>16-3</b>
<b>pdebound and pdegeom reference pages removed</b> .....	<b>16-3</b>

## R2014b

<b>Functions for modular definition of boundary conditions</b> .....	<b>17-2</b>
<b>pdeInterpolant object for solution interpolation</b> .....	<b>17-2</b>

## R2014a

<b>Damping option for hyperbolic solver</b> .....	<b>18-2</b>
---	-------------



## R2013b

<b>Display option in hyperbolic and parabolic solvers</b> .....	<b>19-2</b>
<b>Eigenvalue example</b> .....	<b>19-2</b>

## R2013a

<b>Performance and robustness enhancements in meshing algorithm</b> ....	<b>20-2</b>
<b>New example</b> .....	<b>20-2</b>

## R2012b

<b>Coefficients of parabolic and hyperbolic PDEs that can be functions of the solution and its gradient</b> .....	<b>21-2</b>
<b>Graphics export from pdetool</b> .....	<b>21-2</b>
<b>pdegplot labels edges and subdomains</b> .....	<b>21-2</b>
<b>New examples</b> .....	<b>21-2</b>
<b>pdesmech shear strain calculation change</b> .....	<b>21-2</b>



# R2022b

---

**Version: 3.9**

**New Features**

**Bug Fixes**

## **Interactive Solution Plots: Create and explore visualizations of PDE results**

With the **Visualize PDE Results** Live Editor task, you can plot and inspect results of structural, thermal, electromagnetic, or general PDE analysis using interactive controls. An interactive solution plot enables you to:

- Select which solution data to display.
- Quickly access plots for different frequencies, modes, time steps, and phases.
- Show animated solutions.
- Add a mesh to a solution plot.
- Adjust color limits for a colormap.
- Adjust surface transparency.

## **DC Conduction Analysis: Find stationary current distribution in conductors due to applied voltage**

The programmatic workflow for DC conduction analysis enables you to analyze stationary current distribution in conductors due to applied voltage. Knowing stationary current distributions, you can:

- Study the concentration of current density near sharp geometric corners.
- Perform a magnetostatic analysis with a known current distribution.

For details, see `ConductionResults`.

## **Magnetostatic Analysis: Model permanent magnets**

Specify magnetization using the `electromagneticSource` function to account for materials generating their own magnetic fields in a magnetostatic analysis workflow. For an example, see `MagnetostaticResults`.

## **Geometries: Import 3-D geometries from STEP files**

Import a 3-D geometry in STEP file format using the `importGeometry` function. When importing from a STEP file, you can control the accuracy of the geometry import by specifying the relative sag value. For examples of the STEP files included in the toolbox, see “STEP File Import”.

# R2022a

---

**Version: 3.8**

**New Features**

**Bug Fixes**

**Compatibility Considerations**

## Electromagnetic Analysis: Solve 2-D and 3-D wave scattering and transmission problems

The programmatic workflow for a harmonic electromagnetic analysis enables you to set up, solve, and analyze time-harmonic Maxwell's equations (the Helmholtz equation). This workflow enables you to specify:

- Frequency-dependent relative permittivity, relative permeability, and conductivity of the material
- Frequency and components of the electric or magnetic field on boundaries
- Absorbing boundary conditions

For an example, see `Scattering Problem`.

## Reduced-Order Modeling in Thermal Analysis: Approximate dynamic characteristics of a thermal model for faster execution

Finite-element discretized models can be very large. Model-order reduction techniques approximate the dynamics of the original system with a smaller system while retaining most of the dynamic characteristics.

When using the reduced-order model workflow, you can:

- Compute modes of a thermal model using eigenvalue or proper orthogonal decomposition. For details, see `solve`.
- Speed up computations for a transient thermal model by using the computed modes. For details, see `solve`.
- Generate a reduced-order model by projecting the full finite-element system onto modes. For details, see `reduce`.

## Hysteretic Damping: Model energy loss due to internal friction

Specify hysteretic damping of a material for direct and modal frequency response models. The toolbox enables you to specify a constant hysteretic damping parameter:

- For the entire geometry
- For specified cells of a 3-D geometry or specified faces of a 2-D geometry

For details, see `structuralProperties`.

## Functionality being removed or changed

### **pdecont will be removed**

*Still runs*

`pdecont` will be removed in a future release. Use `pdeplot` instead.

To update your code for plotting the default 10 levels `u`, change instances of the function call `pdecont(p, t, u)` to the function call:

```
pdeplot(p, [], t, "XYData", u, "XStyle", "off", ...  
        "Contour", "on", "ColorBar", "off")
```

---

To update your code for plotting the first  $n$  levels, change instances of the function call `pdecont(p, t, u, n)` to the function call:

```
pdeplot(p, [], t, "XYData", u, "XStyle", "off", ...
         "Contour", "on", "Levels", n, ...
         "ColorBar", "off")
```

To update your code for plotting levels specified as a vector  $v$ , change instances of the function call `pdecont(p, t, u, v)` to the function call:

```
pdeplot(p, [], t, "XYData", u, "XStyle", "off", ...
         "Contour", "on", "Levels", v, ...
         "ColorBar", "off")
```

`pdeplot` gives you more control over your contour plot.

Note that the legacy workflow that uses the `[p, e, t]` mesh is not recommended. New features might not be compatible with this legacy workflow. For a description of the mesh data in the recommended workflow, see Mesh Data.

### **pdesurf will be removed**

*Still runs*

`pdesurf` will be removed in a future release. Use `pdeplot` instead.

To update your code for plotting a solution  $u$  at nodes, change instances of the function call `pdesurf(p, t, u)` to the function call:

```
pdeplot(p, [], t, "XYData", u, "ZData", u, "ColorBar", "off")
```

To update your code for plotting a solution  $ut$  at midpoints of mesh triangles, change instances of the function call `pdesurf(p, t, ut)` to the function call:

```
pdeplot(p, [], t, "XYData", ut, "XStyle", "flat", ...
         "ZData", ut, "ZStyle", "discontinuous", ...
         "ColorBar", "off")
```

`pdeplot` gives you more control over your surface plot.

Note that the legacy workflow that uses the `[p, e, t]` mesh is not recommended. New features might not be compatible with this legacy workflow. For a description of the mesh data in the recommended workflow, see Mesh Data.





# R2021b

---

**Version: 3.7**

**New Features**

**Bug Fixes**

## **Electromagnetic Analysis: Solve 3-D electrostatic and magnetostatic problems**

The programmatic workflow for electromagnetic analysis enables you to set up, solve, and analyze 3-D electrostatic and magnetostatic problems using natural steps and familiar domain-specific language.

For more details about the new workflow, see [Electromagnetics](#).

## **Local Mesh Refinement: Set target mesh sizes around specified vertices, edges, and faces**

The `generateMesh` function lets you specify local target mesh size around geometric vertices, edges, and faces. The function generates a mesh with the element sizes around the specified regions as close to the target sizes as possible, and gradually blends the mesh between regions with different element sizes.

## **Structural and Thermal Analysis: Extract sparse linear models for use with Control System Toolbox**

Use the `linearize` function to obtain `mechss` (Control System Toolbox) and `sparss` (Control System Toolbox) models from structural and thermal models, respectively. You can also use the function to extract the finite element matrices used to obtain the `mechss` and `sparss` models. For examples, see [Linear Analysis of Cantilever Beam](#) (Control System Toolbox) and [Linear Analysis of Tuning Fork](#) (Control System Toolbox).

## **PDEVisualization Properties: Display or hide axes and control background color**

Use `PDEVisualization` Properties to display or hide the axes and also change the background color.

# R2021a

---

**Version: 3.6**

**New Features**

**Bug Fixes**

## Electromagnetic Analysis: Solve 2-D electrostatic and magnetostatic problems

A new programmatic workflow lets you set up, solve, and analyze 2-D electrostatic and magnetostatic problems using natural steps and familiar domain-specific language. When setting up an electromagnetic problem, you can:

- Create a special model container for an electrostatic or magnetostatic model using the `createpde` function.
- Assign electromagnetic material properties, such as relative permittivity and relative permeability, using the `electromagneticProperties` function.
- Specify a charge density and current density using the `electromagneticSource` function.
- Set electromagnetic boundary conditions, such as voltage and magnetic potential, using the `electromagneticBC` function.

The `solve` function solves electrostatic or magnetostatic problems and returns the values of electric or magnetic potential, field, and flux density at nodal locations of the mesh. You can analyze the resulting solution further by using one of these functions:

- `interpolateElectricPotential` and `interpolateMagneticPotential` interpolate the electric and magnetic potential at nodes to arbitrary spatial locations.
- `interpolateElectricField` and `interpolateMagneticField` interpolate the electric and magnetic field at nodes to arbitrary spatial locations.
- `interpolateElectricFlux` and `interpolateMagneticFlux` interpolate the electric and magnetic flux density at nodes to arbitrary spatial locations.

The `pdeplot` function visualizes the solution by plotting the resulting potentials, fields, fluxes, and mesh.

For more details about the new workflow, see [Electromagnetics](#).

## 3-D Geometries: Create a multidomain geometry by inserting one geometry into another

Combine two 3-D geometries or create void regions inside 3-D geometry using the `addCell` and `addVoid` functions. For these operations, all cells of one geometry must be contained inside one cell of the other geometry.

## Geometry Extrusion: Extrude faces of a 3-D geometry to a specified height

Extrude faces of a 3-D geometry along the direction normal to all specified faces using the `extrude` function. You can extrude faces into multiple layers by specifying a vector height of thicknesses of the layers. All faces must be flat and must have the same orientation.

---

## Geometry Queries: Find faces, edges, and cells based on their proximity to other geometric entities

The following functions support geometry queries. They return face and edge IDs that you can use as arguments for other functions.

- `cellFaces` and `cellEdges` find faces and edges attached to the specified cells.
- `faceEdges` finds edges attached to the specified faces.
- `facesAttachedToEdges` finds faces attached to the specified edges.
- `nearestFace` and `nearestEdge` find faces and edges closest to the specified points.

## Geometry Import from STL Files: Import a geometry without creating a model

The `importGeometry` function now lets you create a geometry object from an STL geometry file without attaching the imported geometry to any model. The new syntax is `g = importGeometry('geometryfile.stl')`. This approach lets you modify and adapt the geometry to meet your requirements before attaching it to a model. It also lets you share the same geometry between models.

## PDEVisualization object: Create and plot an object containing nodal data, mesh, and visualization options as its properties

The `pdeviz` function uses the mesh and nodal data to create a `PDEVisualization` object and plot the result. You can interact with the resulting plot by changing the properties of this object. For example, you can change the mesh visibility, transparency, mesh deformation, axes labels, and so on. The plot reflects the new settings without the need to plot the same data repeatedly.

## Face Addition and Geometry Transformation for Analytic Geometries: `addFace`, `rotate`, `scale`, and `translate` now work with `AnalyticGeometry` objects

The `addFace` function now lets you fill void regions in 2-D `AnalyticGeometry` objects.

The `rotate`, `scale`, and `translate` functions now work on `AnalyticGeometry` objects.

## PDE Conversion to Divergence Form: Convert PDEs into the form required by Partial Differential Equation Toolbox using Symbolic Math Toolbox functions

Partial Differential Equation Toolbox solves equations of the form

$$\mathbf{m} \frac{\partial^2 \mathbf{u}}{\partial t^2} + \mathbf{d} \frac{\partial \mathbf{u}}{\partial t} - \nabla \cdot (\mathbf{c} \otimes \nabla \mathbf{u}) + \mathbf{a} \mathbf{u} = \mathbf{f}$$

The `pdeCoefficients` (Symbolic Math Toolbox) and `pdeCoefficientsToDouble` (Symbolic Math Toolbox) functions convert PDEs into this form and extract the coefficients into a structure that can

be used by the `specifyCoefficients` function. For example, see [Solve Partial Differential Equation of Nonlinear Heat Transfer](#) (Symbolic Math Toolbox).

# R2020b

---

**Version: 3.5**

**New Features**

**Bug Fixes**

### **3-D Geometry Creation: Extrude a 2-D geometry into a 3-D geometry**

Extrude a planar geometry along the z-axis to obtain a 3-D geometry. For details, see 3-D Multidomain Geometry from 2-D Geometry and `extrude`.

### **Finite Element Matrices for Custom Workflows: Assemble FE matrices for time-dependent and nonlinear models**

The `assembleFEMatrices` function, which returns global finite element matrices for structural, thermal, and general PDE models can now:

- Assemble a subset of matrices, such as updated load only or stiffness only, to save computation time
- Assemble matrices using input time or solution for a time-dependent or nonlinear model, respectively



# R2020a

---

**Version: 3.4**

**New Features**

**Bug Fixes**

**Compatibility Considerations**

## **Axisymmetric Analysis: Speed up simulations by simplifying 3-D solids of revolution by analyzing only the 2-D axisymmetric sections**

Simplify 3-D structural and thermal problems to 2-D using their symmetry around the axis of rotation. The toolbox now supports an axisymmetric analysis for linear elasticity and heat transfer problems and enables you to:

- Create an axisymmetric 2-D model for a structural or thermal problem. See `createpde`.
- Set up an axisymmetric problem and analyze the results in cylindrical coordinates. In particular, you can specify the enforced displacement for a structural axisymmetric model. See `structuralBC`.
- Model the effect of a spinning axisymmetric structure by specifying a centrifugal load. For details, see `structuralBodyLoad`.

For a heat transfer example, see *Heat Distribution in Circular Cylindrical Rod*.

## **Multidomain Geometry: Split cells and fill voids to create multiple domains with different properties**

Use the `addFace` function to create new geometric faces in 2-D and 3-D geometries. Adding a new face lets you modify a geometry:

- In a 2-D geometry, a new face fills a void region. The new face can have its own properties, such as thermal or structural material properties, or become an internal heat source.
- In a 3-D geometry, a new face splits a cell into two cells. Splitting cells lets you convert a single domain geometry (created by `importGeometry`, for example) to a multidomain geometry. Each cell (domain) can have its own properties.

## **Geometry Transformation: Manipulate geometries to preferred orientation and size by rotation, scaling, and translation**

The toolbox supports these geometry transformations:

- Rotation around an arbitrary axis using the `rotate` function.
- Scaling up or down along each Cartesian axis independently with respect to a reference point using the `scale` function.
- Translation in any direction using the `translate` function.

## **Modal Damping: Include damping in modal transient and frequency response simulations**

For modal transient and modal frequency response models, specify damping as a percentage of critical damping for a selected vibration frequency. The toolbox enables you to specify the modal damping parameter as:

- Constant modal damping ratio for all modes
- Frequency-dependent modal damping ratio using a function handle

---

See `structuralDamping` and Dynamics of Damped Cantilever Beam.

## **pdegplot, pdemesh, pdeplot, and pdeplot3D Functions: Improved performance for plots with many text labels**

The PDE plotting functions show faster rendering and better responsiveness for plots that display many text labels. For example, this code shows about a 3x increase in speed:

```
model = createpde;  
importGeometry(model, 'Block.stl')  
generateMesh(model)  
pdemesh(model, 'NodeLabels', 'on')
```

### **Compatibility Considerations**

Code containing `findobj(fig, 'Type', 'Text')` no longer returns labels on figures produced by the PDE plotting functions.



# R2019b

---

**Version: 3.3**

**New Features**

**Bug Fixes**

## Reduced Order Modeling: Approximate dynamic characteristics of a structural model for faster execution

The toolbox now supports the Craig-Bampton component mode synthesis workflow for structural mechanics.

Finite-element discretized models can be very large. Model order reduction techniques approximate the dynamics of the original system with a smaller system while retaining most of the dynamic characteristics.

When using the reduced-order model workflow, you can:

- Specify structural superelement interfaces. See `structuralSEInterface`.
- Reduce a structural analysis model to all modes in the specified frequency range and the interface degrees of freedom. See `reduce`.

## Simscape Multibody Models: Generate dynamic substructure for flexible multibody simulations

You can use results obtained with the model order-reduction technique to provide data for Simscape™ Multibody™ flexible body block.

Simscape models expect the connections at all joints to have six degrees of freedom, while Partial Differential Equation Toolbox uses two or three degrees of freedom at each node. To interface with Simscape joints, set multipoint constraints by using the `structuralBC` function. Setting a multipoint constraint ensures that all nodes and all degrees of freedom have a rigid constraint with the geometric center of all specified geometric regions together as the reference point. The reference location has six degrees of freedom.

To recover a full-model transient solution from reduced-order model results and Simscape solution, use the `reconstructSolution` function.

For an example of a Simscape Multibody workflow that uses a reduced order model, see [Model an Excavator Dipper Arm as a Flexible Body \(Simscape Multibody\)](#).

## Frequency Response Analysis: Determine dynamic response of a structure in the frequency domain

Determining the dynamic response of a structure under various forms of excitation is a common task in structural analysis. This problem can be formulated over time domain (in which case it is called transient response analysis) or frequency domain (called frequency response analysis or sometimes harmonic analysis).

In addition to the transient dynamic analysis, the toolbox now supports the frequency response analysis for structural models. When setting up a linear elasticity problem, you can:

- Create a special model container for a frequency response structural analysis. See `createpde`.
- Specify a boundary load, for example, pressure. See `structuralBoundaryLoad`.
- Solve the problem and obtain displacement, velocity, acceleration, and solution frequencies at nodal locations of the mesh. See `solve`.

- 
- To speed up computations, you can use modal analysis results for frequency response analysis. The `ModalResults` argument triggers the `solve` function to use the modal superposition method.

## **Structural Analysis: Set concentrated boundary conditions at arbitrary locations on geometry surfaces**

Use `addVertex` to create new vertices at any points on boundaries of a 2-D or 3-D geometry represented by a `DiscreteGeometry` object. Then use `structuralBC` or `structuralBoundaryLoad` to set concentrated boundary constraints and loads at these vertices.

For an example of applying a point load on a geometric boundary, see [Reduced-Order Modeling Technique for Beam with Point Load](#).

## **Eigenvalue Solver Options: Specify parameters for Lanczos algorithm**

Solvers for structural modal analysis problems and reduced-order modeling use the Lanczos algorithm. Use `PDESolverOptions` to specify the maximum number of Lanczos shifts and the block size for block Lanczos recurrence.





# R2019a

---

**Version: 3.2**

**New Features**

**Bug Fixes**

## **Structural Analysis: Set boundary constraints and loads on edges and vertices**

The workflow for structural analysis problems lets you specify an enforced displacement and the 'free' and 'fixed' boundary constraints on vertices and edges for both 2-D and 3-D models. To specify boundary constraints and a displacement, including spatial components of displacement, use the `structuralBC` function. The previous versions of this function let you specify the displacement and boundary constraints values only on edges for 2-D models and faces on 3-D models.

You can specify concentrated force at a vertex by using the `structuralBoundaryLoad` function.

## **Structural Analysis: Compute transient response by using the modal superposition method**

By default, the `solve` function uses the direct integration approach which can be slow. To speed up calculations, transient dynamics analysis can be performed using modal analysis results. The new `ModalResults` argument triggers the `solve` function to switch to the modal transient solver instead of using the direct integration approach.

## **Finite Element Matrices: Assemble global finite element matrices for thermal and structural models**

The `assembleFEMatrices` function accepts thermal and structural models with time- and solution-independent coefficients.

# R2018b

---

**Version: 3.1**

**New Features**

**Bug Fixes**

## Thermal Load: Evaluate thermal stress and thermal expansion in static structural analysis

The programmatic workflow for static structural analysis problems now lets you account for thermal loading. When setting up a static structural problem, you can:

- Specify a coefficient of thermal expansion as a structural material property. See `structuralProperties`.
- Specify a thermal load as a body load. Use a constant temperature to specify a steady and uniform thermal load or use previously obtained results of thermal analysis to specify a nonuniform thermal load. See `structuralBodyLoad`.
- Specify a reference temperature. This temperature corresponds to the state of the model at which both thermal stress and strain are zeros. See `StructuralModel`.

The `solve` function solves a static structural analysis model accounting for mechanical and thermal effects. The function returns a displacement, stress, strain, and von Mises stress induced by both mechanical and thermal loads.

For an example of the new workflow, see [Thermal Deflection of Bimetallic Beam](#).

# R2018a

---

**Version: 3.0**

**New Features**

**Bug Fixes**

**Compatibility Considerations**

## Structural Analysis: Solve linear transient dynamic problems using direct time integration

The programmatic workflow for structural analysis problems now lets you set up, solve, and analyze dynamic linear elasticity problems using a familiar domain-specific language. Most steps correspond to the similar steps for setting up a static linear elasticity problem. Additional options for a dynamic linear elasticity problem include:

- Specifying proportional (Rayleigh) mass and stiffness damping parameters. See `structuralDamping`.
- Specifying initial conditions. See `structuralIC`.
- Specifying time-dependent boundary loads by using function handles. Also, you can specify the form and duration of the pressure pulse and the frequency and phase of sinusoidal pressure by using the corresponding name-value pair arguments. See `structuralBoundaryLoad`.
- Specifying time-dependent boundary conditions by using function handles. Also, you can specify the form and duration of  $x$ -,  $y$ -, or  $z$ -component of the enforced displacement and the frequency and phase of sinusoidal displacement by using the corresponding name-value pair arguments. See `structuralBC`.

The `solve` function solves a dynamic structural analysis model and returns displacement, velocity, and acceleration at nodal locations of the mesh. You can analyze the resulting solution further by using one of these functions:

- `evaluateStress` evaluates stress at nodal locations for all time steps.
- `evaluateStrain` evaluates strain at nodal locations for all time steps.
- `evaluateVonMisesStress` evaluates von Mises stress at nodal locations for all time steps.
- `evaluateReaction` evaluates reaction forces on a specified boundary.
- `evaluatePrincipalStress` and `evaluatePrincipalStrain` evaluate principal stress and strain at mesh nodes.
- `interpolateDisplacement` interpolates displacements at arbitrary spatial locations.
- `interpolateVelocity` interpolates velocities at arbitrary spatial locations.
- `interpolateAcceleration` interpolates accelerations at arbitrary spatial locations.
- `interpolateStress` and `interpolateStrain` interpolate stress and strain at arbitrary spatial locations.
- `interpolateVonMisesStress` interpolates von Mises stress at arbitrary spatial locations.

For more details about the new workflow, see Structural Mechanics.

## Modal Analysis: Find natural frequencies and mode shapes

The programmatic workflow for modal analysis problems lets you find natural frequencies and mode shapes of a structure. Use the same steps for creating a modal analysis model and specifying materials and boundary constraints as for static linear elasticity problems. When solving a modal analysis model, the solver requires a frequency range parameter and returns the modal solution in that frequency range. See Vibration of Square Plate.

---

## Mesh Analysis: Assess element quality, measure element area or volume, and find nodes and elements

The `findElements` and `findNodes` functions return collections of mesh elements and nodes associated with a particular geometric region (edge, face, and so on), located within a bounding box or disk, or located close to a specified point or node. These functions return node or element IDs that you can use as arguments for other mesh query functions or plotting functions.

The `meshQuality` function lets you assess shape quality of mesh elements of an entire mesh or a particular group of elements. Using this function, you can analyze the shape quality of each element of a 2-D or 3-D mesh, locate the elements of low quality, and adjust your mesh-generation parameters accordingly.

The `area` and `volume` functions measure areas and volumes covered by individual elements and groups of elements of 2-D and 3-D meshes, respectively.

The `pdemesh`, `pdeplot`, and `pdeplot3D` functions accept node and element IDs as input arguments, which lets you highlight particular nodes and elements on mesh plots.

## Geometry from Mesh: Create multidomain geometry from nodes and elements

The `geometryFromMesh` function creates a geometric model from a triangulated mesh, including planar, volume, and surface triangulation that bounds a closed volume. This function lets you create a multidomain geometry by specifying the subdomain ID for each element of the mesh. You can retain the original mesh or generate a new one by using the `generateMesh` function.

## Functionality Being Removed or Changed

Functionality	Result	Use Instead	Compatibility Considerations
<code>pdesdp</code>	Still runs	<code>findNodes</code>	Use a <code>PDEModel</code> object to set up and solve a PDE problem. For the step-by-step workflow, see <a href="#">Solve Problems Using PDEModel Objects</a> . <code>PDEModel</code> stores a finite element mesh as a <code>FEMesh</code> object in its <code>Mesh</code> property. Use the <code>findNodes</code> function to find mesh nodes that belong to particular regions.
<code>pdesdt</code>	Still runs	<code>findElements</code>	Use <code>findElements</code> to find mesh elements that belong to particular regions.
<code>pdetrg</code>	Still runs	<code>area</code>	Use the <code>area</code> function to compute areas of 2-D mesh elements.
<code>pdetriq</code>	Still runs	<code>meshQuality</code>	Use the <code>meshQuality</code> function to evaluate shape quality of mesh elements.

<b>Functionality</b>	<b>Result</b>	<b>Use Instead</b>	<b>Compatibility Considerations</b>
pdegrad	Still runs	evaluateGradient	Use the evaluateGradient function to evaluate the gradient of a PDE solution.
pdecgrad	Still runs	evaluateCGradient	Use the evaluateCGradient function to evaluate the flux of a PDE solution.



# R2017b

---

**Version: 2.5**

**New Features**

**Bug Fixes**

**Compatibility Considerations**

## Structural Analysis: Solve static linear elasticity problems

A new programmatic workflow lets you set up, solve, and analyze static linear elasticity problems using a familiar domain-specific language. When setting up a static linear elasticity problem, you can:

- Create a special model container for a static structural analysis. See `createpde`.
- Assign structural properties of the material, such as Young's modulus, Poisson's ratio, and mass density. See `structuralProperties`.
- Specify a body load for a structural model. See `structuralBodyLoad`.
- Specify pressure, surface traction, and translational stiffness for a particular boundary. See `structuralBoundaryLoad`.
- Specify enforced displacement and boundary constraints for a particular boundary. See `structuralBC`.

The `solve` function solves a static structural analysis model and returns displacement, stress, strain, and von Mises stress at nodal locations of the mesh. You can analyze the resulting solution further by using one of these functions:

- `interpolateDisplacement` interpolates displacements at arbitrary spatial locations.
- `interpolateStress` and `interpolateStrain` interpolate stress and strain at arbitrary spatial locations.
- `interpolateVonMisesStress` interpolates von Mises stress at arbitrary spatial locations.
- `evaluateReaction` evaluates reaction forces on a specified boundary.
- `evaluatePrincipalStress` and `evaluatePrincipalStrain` evaluate principal stress and strain at mesh nodes.

The `pdeplot` and `pdeplot3D` functions let you visualize the solution by plotting the resulting displacements, stresses, and strains.

For more details about the new workflow, see Structural Mechanics.

## Planar STL Geometry: Import and mesh planar STL geometries

`importGeometry` can import a planar STL geometry and convert it to a 2-D geometry by mapping it to the X-Y plane. To mesh the resulting 2-D geometry, use `generateMesh`.

## Meshing: Improved mesh generation

The mesh generator, `generateMesh`, now:

- Uses new mesh generation algorithm for 2-D geometries.
- Lets you specify mesh growth rate for both 2-D and 3-D meshes. To specify mesh growth rate, use the `Hgrad` argument of `generateMesh`.

## Compatibility Considerations

Resulting meshes can differ from meshes generated in previous releases. For example, meshes generated with the default size controls can have fewer elements than before.

Also, `generateMesh` creates quadratic meshes for 2-D problems by default. In previous releases, the default mesh for 2-D geometries is a linear mesh. For both 2-D and 3-D geometries, you can specify whether you want to use linear or quadratic mesh by using the `GeometricOrder` argument of `generateMesh`.

## Functionality Being Removed or Changed

Functionality	Result	Use Instead	Compatibility Considerations
Function handle for specifying nonconstant boundary conditions and coefficients of the form <code>@f(problem, region, state)</code>	Still runs	<code>@f(region, state)</code>	Use the two-argument form, <code>@f(region, state)</code> , to define a function handle for <code>applyBoundaryCondition</code> or <code>specifyCoefficients</code> .
<code>pdetool</code>	Still runs	<code>pdeModeler</code>	The function name <code>pdetool</code> has been changed to <code>pdeModeler</code> .
Opening the PDE Modeler app by calling <code>pdeModeler</code> or <code>pdetool</code> or by using the <b>Apps</b> tab	Does not overwrite existing information in the app anymore	To overwrite the existing information in the PDE Modeler app, select <b>File &gt; New</b> .	In previous releases, starting the PDE Modeler app and then reopening the app by calling <code>pdetool</code> or by using the <b>Apps</b> tab overwrites any existing information in the app. Now, reopening the PDE Modeler app brings focus to the app window.
<code>Jiggle</code> , <code>JiggleIter</code> , and <code>MeshVersion</code> arguments of <code>generateMesh</code>	Ignored	No replacement	<code>generateMesh</code> produces good quality meshes without jiggling the nodes.



# R2017a

---

**Version: 2.4**

**New Features**

**Bug Fixes**

**Compatibility Considerations**

## Thermal Analysis: Solve conduction-dominant heat transfer problems with convection and radiation occurring at boundaries

A new programmatic workflow lets you set up, solve, and analyze conduction-dominant heat transfer problems using natural steps and familiar domain-specific language. When setting up a heat transfer problem, you can:

- Create a special model container for a steady-state or transient thermal model. See `createpde`.
- Assign thermal properties of the material, such as thermal conductivity, specific heat, and mass density. See `thermalProperties`.
- Specify internal heat sources of the geometry. See `internalHeatSource`.
- Set temperatures on boundaries, specify insulated boundaries, heat fluxes through boundaries, convection coefficients, radiation emissivity coefficients, and ambient temperature. See `thermalBC`.
- Set initial temperature or initial guess for temperature. See `thermalIC`.

The `solve` function solves steady-state and transient thermal models and returns temperatures and temperature gradients at nodal locations of the thermal model mesh. You can analyze the resulting solution further by using one of these functions:

- `interpolateTemperature` interpolates resulting temperatures to arbitrary spatial locations.
- `evaluateTemperatureGradient` evaluates temperature gradient for a transient thermal solution at arbitrary spatial locations.
- `evaluateHeatFlux` evaluates heat flux of a thermal solution at nodal or arbitrary spatial locations.
- `evaluateHeatRate` evaluates integrated heat flow rate normal to specified boundary.

The `pdeplot` and `pdeplot3D` functions let you visualize a thermal model solution by plotting the resulting temperatures, and temperature gradients, and mesh.

There is a new featured example showing heat transfer in a transient 3-D model, Heat Conduction in a Spherical Multidomain Geometry with Nonuniform Heat Flux.

For more details about the new workflow, see Heat Transfer.

## 3-D Multidomain Geometry: Create geometries consisting of stacked or nested cuboids, spheres, or cylinders

The `multicuboid`, `multicylinder`, and `multisphere` functions create 3-D geometries formed by several cubic, cylindrical, and spherical cells, respectively. With these functions, you can create stacked or nested geometries. You also can create geometries where some cells are empty, for example, hollow cylinders, cubes, or spheres.

All cells in a geometry must be of the same type: either cuboids, or cylinders, or spheres. These functions do not combine cells of different types in one geometry.

---

## Functionality Being Removed or Changed

Functionality	Result	Use Instead	Compatibility Considerations
pdemdlcv	Errors	No replacement	Remove all instances of pdemdlcv.
Function handle for specifying nonconstant boundary conditions and coefficients of the form <code>@f(problem, region, state)</code> .	Still runs	<code>@f(region, state)</code>	Use the two-argument form, <code>@f(region, state)</code> , to define a function handle for <code>applyBoundaryCondition</code> or <code>specifyCoefficients</code> .





# R2016b

---

**Version: 2.3**

**New Features**

**Bug Fixes**

**Compatibility Considerations**

## Flux of PDE Solution: Evaluate tensor product of c-coefficient and gradient of PDE solution for 2-D and 3-D problems

The `evaluateCGradient` function computes the tensor product of c-coefficient and gradients of the PDE solution,  $c \otimes \nabla u$ , at nodal or arbitrary locations in 2-D or 3-D geometry.

## Boundary Conditions: Set an individual boundary condition for each equation in a PDE system, query boundary condition assignments

The `applyBoundaryCondition` function can set an individual boundary condition for each equation in a system of PDEs in one function call. For example, in a system of two PDEs, one `applyBoundaryConditions` call can set the Dirichlet boundary condition for the first equation, and the Neumann boundary condition for the second equation.

The function uses clear precedence rules to decide which condition to use for every geometric region.

The `findBoundaryConditions` function returns the currently used boundary condition assignment for a geometric region.

## Compatibility Considerations

The new syntax requires explicit specification of the boundary condition type as `'dirichlet'`, `'neumann'`, or `'mixed'`. For example, in previous releases you used the following syntaxes to apply the Dirichlet boundary condition on edge 1 and Neumann boundary condition on face 2.

```
applyBoundaryCondition(model,'Edge',1,'u',0);  
applyBoundaryCondition(model,'Face',2,'g',0,'q',0);
```

Update them to specify the boundary condition types.

```
applyBoundaryCondition(model,'dirichlet','Edge',1,'u',0);  
applyBoundaryCondition(model,'neumann','Face',2,'g',0,'q',0);
```

Previously, to set the Dirichlet boundary condition for one equation in a PDE system and the Neumann boundary condition for another, you used two function calls to `applyBoundaryCondition`.

```
applyBoundaryCondition(model,'edge',4,'u',0,'EquationIndex',1);  
applyBoundaryCondition(model,'edge',4,'g',[0,-1]);
```

Now, set both boundary conditions in one function call by specifying the boundary condition type as `'mixed'`.

```
applyBoundaryCondition(model,'mixed','edge',4,'u',0,...  
                        'EquationIndex',1,'g',[0,-1]);
```

## Nodal Initial Conditions: Set nodal initial conditions using the result of a previous simulation

The `setInitialConditions` function sets initial conditions at the mesh nodes by using the solution from a previous analysis on the same geometry and mesh.

---

## Geometric Initial Conditions: Set initial conditions on vertices, edges, faces, and cells

The `setInitialConditions` function sets initial conditions on faces, edges, and vertices for both 2-D and 3-D geometries, and on cells for 3-D geometry. You can set initial conditions on particular vertices, edges, faces, and cells or set global initial conditions on all vertices, edges, faces, or cells in the geometry.

## Mesh and Solution Plots: Set transparency, display node and element labels

The `FaceAlpha` argument of `pdegplot`, `pdemesh`, `pdeplot`, and `pdeplot3D` lets you set the plot transparency.

The `ElementLabels` argument of `pdemesh`, `pdeplot`, and `pdeplot3D` lets you display element labels.

The `NodeLabels` argument of `pdemesh` and `pdeplot` lets you display node labels.

The `VertexLabels` and `CellLabels` arguments of `pdegplot` let you display vertex and cell labels.

## Compatibility Considerations

The argument `SubdomainLabels` is no longer recommended. Use `FaceLabels` for 2-D geometries instead.

## 3-D Geometry Import from STL Files: Improved quality of the resulting geometry

When importing an STL geometry for a 3-D problem, `importGeometry` can recognize and reconstruct more geometric vertices, edges, and faces of the original CAD geometry in some instances. In these cases, the resulting geometry is a closer match to the original CAD geometry.

## Compatibility Considerations

Detailed geometries can now contain more faces and edges than in previous releases. As a result, in rare instances, the new faces and edges can cause renumbering of the existing ones. If your code imports an STL geometry, visually check the geometry to ensure that you are assigning boundary and initial conditions to the intended regions.

## Functionality Being Removed or Changed

Functionality	Result	Use Instead	Compatibility Considerations
<code>pdemdlcv</code>	Warns	No replacement	Remove all instances of <code>pdemdlcv</code> .

Functionality	Result	Use Instead	Compatibility Considerations
pde	Errors	createpde	Use createpde to create a PDEModel that holds the PDE analysis data.  The pde class was a value class. The replacement PDEModel class is a handle class.
pdeGeometryFromEdges	Errors	geometryFromEdges	Use geometryFromEdges instead.  Although pdeBoundaryConditions still runs with a warning, its returned type has changed from a pdeGeometry object to an AnalyticGeometry object. The pdeGeometry class was a value class. The replacement AnalyticGeometry class is a handle class.
pdeBoundaryConditions	Errors	applyBoundaryCondition	Replace all instances of pdeBoundaryConditions(ApplicationRegion,...) with applyBoundaryCondition(model,BCType,'edge',EdgeID,...)  Although pdeBoundaryConditions still runs with a warning, its returned type has changed from a pdeBoundaryConditions object to a 1BoundaryCondition object. The pdeBoundaryConditions class was a value class. The replacement BoundaryCondition class is a handle class.
Function handle for specifying nonconstant boundary conditions and coefficients of the form @f(problem,region,state).	Still runs	@f(region,state)	Use specifyCoefficients and define a function handle that takes only two arguments:  @f(region,state)

# R2016a

---

**Version: 2.2**

**New Features**

**Bug Fixes**

**Compatibility Considerations**

## **PDE Solvers: Use `solvepde` and `solvepdeeig` functions to solve PDEs and PDE eigenvalue problems**

New solver functions for `PDEModel`:

- `solvepde` replaces `asempde`, `pdenonlin`, `hyperbolic`, and `parabolic`.
- `solvepdeeig` replaces `pdeeig`.

To use the new solvers, include PDE coefficients in your model using `specifyCoefficients`. Include initial conditions in your model using `setInitialConditions`.

The new solvers return results as one of the three new objects:

- `StationaryResults` — Returned by `solvepde` for a stationary PDE model.
- `TimeDependentResults` — Returned by `solvepde` for a time-dependent PDE model.
- `EigenResults` — Returned by `solvepdeeig`.

`StationaryResults` and `TimeDependentResults` objects contain solution gradients at the nodes.

## **PDE Coefficients: Specify equation coefficients as a property of `PDEModel`**

The `specifyCoefficients` function specifies equation coefficients as a property of `PDEModel`.

## **Initial Conditions: Set initial conditions or initial guess as a property of `PDEModel`**

The `setInitialConditions` function specifies initial conditions as a property of `PDEModel`.

## **Quadratic Elements for 2-D Mesh: Generate 2-D mesh using quadratic triangular elements**

Create a quadratic mesh for 2-D problems using `generateMesh` with `GeometricOrder` set to `'quadratic'`.

## **Gradient of PDE Solution: Evaluate solution gradient at arbitrary 2-D or 3-D points**

The `evaluateGradient` function enables you to interpolate the gradient of a `StationaryResults` or `TimeDependentResults` object at arbitrary points in the geometry.

## **Finite Element Matrices: Use `assembleFEMatrices` to assemble finite element matrices**

The `assembleFEMatrices` function assembles finite element matrices for independent factoring and solution using linear algebra methods. It replaces `asempde`, `assemma`, and `assemb` for matrix assembly.

---

## PDE Results for Plotting and Postprocessing: New result objects depend on the type of PDE

The `createPDEResults` function returns results as one of the three new objects, depending on the type of the PDE problem.

- A `StationaryResults` object for a stationary PDE model. `StationaryResults` contains the solution of PDE and its gradients at the nodal locations.
- A `TimeDependentResults` object for a time-dependent PDE model. `TimeDependentResults` contains the solution of PDE and its gradients at the nodal locations.
- A `EigenResults` object for an eigenvalue problem.

## Compatibility Considerations

`createPDEResults` no longer creates an object of type `PDEResults`.

The syntax of `createPDEResults` has changed to accommodate creating the new result types for time-dependent and eigenvalue problems.

- To create the `TimeDependentResults` object for a time-dependent problem, use the syntax `createPDEResults(pdem,u,utimes,'time-dependent')`, where `utimes` is a vector of solution times.
- To create the `EigenResults` object for an eigenvalue problem, use the syntax `createPDEResults(pdem,eigenvectors,eigenvalues,'eigen')`.

`EigenResults` has different property names than `PDEResults`. Update any eigenvalue scripts that use `PDEResults` property names.

## Functionality Being Removed or Changed

Functionality	Result	Use Instead	Compatibility Considerations
<code>wbound</code>	Still runs	No replacement	New features and the recommended workflow are not compatible with this function.  For the recommended workflow, see <a href="#">Solve Problems Using PDEModel Objects</a> .
<code>pdeadgsc</code> and <code>pdeadworst</code>	Still runs	No replacement	New features and the recommended workflow are not compatible with these functions.  For the recommended workflow, see <a href="#">Solve Problems Using PDEModel Objects</a> .

Functionality	Result	Use Instead	Compatibility Considerations
asempde, assema, assemb, hyperbolic, parabolic, pdenonlin	Still runs	solvepde and assembleFEMatrices	To solve PDE problems, use solvepde. To obtain finite element matrices, use assembleFEMatrices.  For the recommended workflow, see Solve Problems Using PDEModel Objects.
pdeeig and sptarn	Still runs	solvepdeeig	To solve PDE eigenvalue problems, use solvepdeeig.  For the recommended workflow, see Solve Problems Using PDEModel Objects.
poimesh, poiasma, poicalc, poiindex, poisolv	Still runs	solvepde	To solve Poisson's equations, use solvepde. For details, see Solve Problems Using PDEModel Objects.
pdejmps	Still runs	No replacement	New features and the recommended workflow are not compatible with this function.  For the recommended workflow, see Solve Problems Using PDEModel Objects.
pdesmech	Still runs	PDE app	Use the PDE app instead of pdesmech.
dst and idst	Still runs	No replacement	Remove all instances of dst and idst.
tri2grid and pdeintrp	Still runs	interpolateSolution	Use the interpolation function interpolateSolution provided by StationaryResults, TimeDependentResults and EigenResults.
pdeprtni	Still runs	No replacement	NodalSolution is a property of StationaryResults and TimeDependentResults.  Eigenvectors is the corresponding property of EigenResults.
pdemdlcv	Still runs	No replacement	Remove all instances of pdemdlcv.
pde	Warns	createpde	Use createpde to create a PDEModel that holds the PDE analysis data.  The pde class was a value class. The replacement PDEModel class is a handle class.



Functionality	Result	Use Instead	Compatibility Considerations
pdeGeometryFromEdges	Warns	geometryFromEdges	Use geometryFromEdges instead.  Although pdeBoundaryConditions still runs with a warning, its returned type has changed from a pdeGeometry object to an AnalyticGeometry object. The pdeGeometry class was a value class. The replacement AnalyticGeometry class is a handle class.
pdeBoundaryConditions	Warns	applyBoundaryCondition	Replace all instances of pdeBoundaryConditions(ApplicationRegion,...) with applyBoundaryCondition(model,'edge',EdgeID,...)  Although pdeBoundaryConditions still runs with a warning, its returned type has changed from a pdeBoundaryConditions object to a BoundaryCondition object. The pdeBoundaryConditions class was a value class. The replacement BoundaryCondition class is a handle class.
Loading a pdeBoundaryConditions object from an R2014b MAT file.	Errors	applyBoundaryCondition	Recreate the Boundary Conditions using applyBoundaryCondition.
Function handle for specifying nonconstant boundary conditions and coefficients of the form @f(problem, region, state).	Still runs	@f(region, state)	Use specifyCoefficients and define a function handle that takes only two arguments:  @f(region, state)



# R2015b

---

**Version: 2.1**

**New Features**

**Bug Fixes**

**Compatibility Considerations**

### 3-D geometry creation from a finite element mesh using the `geometryFromMesh` function

The `geometryFromMesh` function creates 3-D geometry from a finite element mesh, or from a triangulated surface mesh. For details, see the function reference page or [Create and View 3-D Geometry](#).

### Data structure to represent solutions using the `createPDEResults` function

The `createPDEResults` function converts a PDE solution into a `PDEResults` object. The `PDEResults` object allows you to interpolate the solution using `interpolateSolution`. For details, see the reference pages.

### Functionality Being Removed or Changed

Functionality	Result	Use Instead	Compatibility Considerations
<code>pdesmech</code>	Still runs	PDE app	Use the PDE app instead of <code>pdesmech</code>

# R2015a

---

**Version: 2.0**

**New Features**

**Bug Fixes**

**Compatibility Considerations**

## 3-D finite element analysis

You can now solve partial differential equations with 3-D geometry. To do so, there is a new workflow that combines the geometry, mesh, and boundary conditions into a `PDEModel` object. You can also use this workflow for 2-D geometry. For details, see [Solve Problems Using PDEModel Objects](#).

## Equation coefficients and boundary conditions for 3-D problems

To specify problem coefficients or boundary conditions in 3-D geometry, you can use strings with a syntax similar to that of a 2-D problem. There is a new way of writing functions for coefficients in 3-D geometries. For details, see [PDE Coefficients and Boundary Conditions](#).

## Compatibility Considerations

To accommodate both 2-D and 3-D geometry, the format of boundary condition objects changed from that introduced in R2014b. The new object is `BoundaryCondition Properties`, and calling `pdeBoundaryConditions` now warns that it will be removed in a future release. If you saved a `pdeBoundaryConditions` object in an R2014b-format MAT file, then loading that file in R2015a can produce an error. Additionally, the syntax for specifying nonconstant boundary conditions has changed. Functions written in the previous syntax continue to work for now.

R2014b Syntax	R2015a Syntax
<code>function bcMatrix = myfun(problem, region, state)</code>	<code>function bcMatrix = myfun(region, state)</code>

For details, see [Changes to Boundary Conditions Object From R2014b](#).

## Elliptic, parabolic, hyperbolic, nonlinear, eigenvalue solvers for 3-D problems

The main toolbox solvers now support problems with 3-D geometry. For a listing of functions that do or do not support 3-D geometry, see [Functions That Support 3-D Geometry](#). Solvers take a `model` argument instead of the previous `b`, `p`, `e`, `t` arguments. For details, see the function reference pages.

## 3-D geometry import from STL files

Import the geometry for a 3-D problem in the STL file format using the `importGeometry` function. For details, see [Create and View 3-D Geometry](#).

## 3-D unstructured meshing using tetrahedra

Create finite element meshes using the `generateMesh` function. For 3-D geometry, the meshes consist of tetrahedra. See [Mesh Data for \[p,e,t\] Triples: 3-D](#).

## Plot function to inspect 3-D solutions

The `pdeplot3D` function plots solutions on the boundaries of 3-D geometry. For details, see [Plot 3-D Solutions](#).

---

## Featured examples with 3-D geometry

There are two new featured examples related to linear elasticity that have 3-D geometry:

- Deflection Analysis of a Bracket
- Vibration of a Square Plate

There is also a new example of plotting slices through a 3-D solution: Contour Slices Through a 3-D Solution.

To run the examples at the MATLAB® command line:

```
echodemo StrainedBracketExample  
echodemo Eigenvaluesofa3DPlateExample  
echodemo ContourSlices3DExample
```

## pdebound and pdegeom reference pages removed

The pdebound and pdegeom reference pages have been replaced by the Boundary Conditions and 2-D Geometry documentation categories.





# R2014b

---

**Version: 1.5**

**New Features**

**Bug Fixes**

## **Functions for modular definition of boundary conditions**

To specify PDE boundary conditions in a modular fashion, per edge or set of edges, use a `pdeBoundaryConditions` specification. For details, see [Steps to Specify a Boundary Conditions Object](#).

## **`pdeInterpolant` object for solution interpolation**

Interpolate a PDE solution to a set of points using `evaluate` on an interpolant. Create the interpolant using `pdeInterpolant`.

# R2014a

---

**Version: 1.4**

**New Features**

**Bug Fixes**

## **Damping option for hyperbolic solver**

You can include damping in the hyperbolic solver in matrix form. There is a new example of dynamics of a damped cantilever beam that shows how to use this feature.

# R2013b

---

**Version: 1.3**

**New Features**

**Bug Fixes**

## Display option in hyperbolic and parabolic solvers

You can disable the display of internal ODE solution details that the hyperbolic and parabolic solvers report. To disable the display, set the `Stats` name-value pair to `'off'`.

## Eigenvalue example

There is a new example of eigenvalues of a circular membrane. View the example [here](#). To run the example at the MATLAB command line:

```
echodemo eigsExample
```

# R2013a

---

**Version: 1.2**

**New Features**

**Bug Fixes**

## Performance and robustness enhancements in meshing algorithm

The meshing (geometry triangulation) functions in `initmesh` and `adaptmesh` provide an enhancement option for increased meshing speed and robustness. Choose the enhanced algorithm by setting the `MesherVersion` name-value pair to `'R2013a'`. The default `MesherVersion` value of `'preR2013a'` gives the same mesh as previous toolbox versions.

The enhancement is available in `pdetool` from the **Mesh > Parameters > Mesher version** menu.

### New example

There is a new example of heat distribution in a radioactive rod. View the example [here](#). To run the example at the MATLAB command line:

```
echodemo radioactiveRod
```



# R2012b

---

**Version: 1.1**

**New Features**

**Compatibility Considerations**

## Coefficients of parabolic and hyperbolic PDEs that can be functions of the solution and its gradient

You can now solve parabolic and hyperbolic equations whose coefficients depend on the solution  $u$  or on the gradient of  $u$ . Use the `parabolic` or `hyperbolic` commands, or solve the equations using `pdetool`. For details, see the function reference pages.

## Graphics export from pdetool

You can save the current `pdetool` figure in a variety of image formats. Save the figure using the **File** > **Export Image** menu. See File Menu.

## pdegplot labels edges and subdomains

`pdegplot` now optionally labels:

- The edges in the geometry
- The subdomains in the geometry

To obtain these labels, set the `edgeLabels` or `subdomainLabels` name-value pairs to 'on'. For details, see the `pdegplot` reference page.

## New examples

There is a new example of uniform pressure load on a thin plate. View the example [here](#). To run the example at the MATLAB command line:

```
echodemo clampedSquarePlateExample
```

There is a new example of nonlinear heat transfer in a thin plate. View the example [here](#). To run the example at the MATLAB command line:

```
echodemo heatTransferThinPlateExample
```

There is a new example of a system of coupled PDEs. View the example [here](#). To run the example at the MATLAB command line:

```
echodemo deflectionPiezoelectricActuator
```

## pdesmech shear strain calculation change

The `pdesmech` function now calculates shear strain according to the engineering shear strain definition. This has always been the documented behavior. However, the previous calculation was performed according to the tensor shear strain calculation, which gives half the value of the engineering shear strain.

## Compatibility Considerations

`pdesmech` now returns shear strain values exactly twice as large as before.