


3. Preprocessing of ABAQUS/CAE

3.1 Create new model database


A finite element analysis in ABAQUS/CAE starts from “ create new model database” in the toolbar. Then save it with a name user defined.


To build finite element model viz. geometry modeling is an important and also the start of FEA. The geometry modeling in ABAQUS/CAE includes Sketch Module and Part Module.


3.2 Sketch Module





In order to create your 3 dimensional model, you need draw your 2 dimensional sketch firstly, then use the 3D features to build it.

Once you click the icon  (create sketch) in **Sketch Module**, a sketch interface will appear. The Sketch is based on three mutually perpendicular planes in the Cartesian coordinate system (XY, YZ, and XZ planes). The drawing toolbox is used to draw sketch.


 **Create isolated point:** using the cursor to create one point.

 **Create lines: connected:** drawing a series of connected lines.

 **Create circle: center and perimeter:** draw a circle by two points, one is the center point, the other is the point on the circle.

 **Create lines: rectangle (4 lines):** draw the rectangle using two diagonally points.

 **Create Ellipse:** Center and perimeter: draw an ellipse by three points.

 **Create Arc:** Tangent to Adjacent Curve

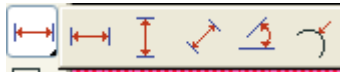
 **Create Arc:** Center and 2 Endpoints

 **Create Arc:** Thru 3 points

 **Create Fillet:** between 2 Curves

 **Create Spline:** Through Pointes

 **Create Construction Line**



Create Dimension




Edit Dimension Value



Edit Vertex Location

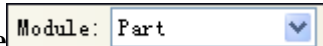
More details about the toolbox in Sketch Module will be presented during ABAQUS training. However, you just need move the cursor to the icon, the help information will be prompted. Then if you want to learn more about this special icon, enter the information to the **search** function in **ABAQUS Documentation**.

After you finish drawing sketch, you need **save** it  for next module—**Part Module**


Tip:

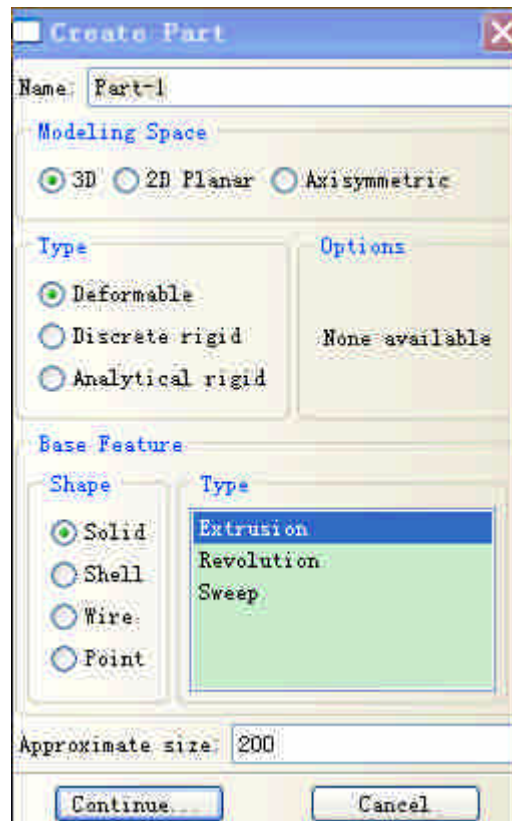
- ✓ You can also use the sketch you've already had in other software, such as CAD. The procedure to import the sketch from the outside of ABAQUS is **File→Import→Sketch**.
- ✓ More details see *19. The Sketch module of ABAQUS/CAE User's Manual in ABAQUS Documentation*


3.3 Part Module




In **Part Module**, the 3D model is created by using the modeling feature such as extrude.

Firstly,  **create part** icon is used to create a new part. It can be repeated to create many different parts.



After click continue button, you will enter the 2D sketch again. Now you have two choices to create your required sketch: **draw it again** or click  to open a **existing sketch**.

Important 3D features


1. **Extrude**  is to create an extruded feature.

Extrusion Distance:


Blend: Specify the distance over which ABAQUS/CAE extrudes the sketch.

Up to face: Select a single face to which ABAQUS/CAE extrudes the sketch.

Through all: This method is available only for extruded cut features


2. **Revolve**  is to create a revolved feature.

You add a revolved solid feature by sketching a **two-dimensional cross-section** and a **construction line** on a selected face. The construction line serves as an axis of revolution, and ABAQUS/CAE creates the solid feature by rotating the cross-section about the axis using a specified angle of revolution.

- 3 **Sweep**  is to create a sweep feature

You add a swept solid feature by sketching a **sweep path** on a selected face and sketching a **sweep profile**. The sweep profile is always perpendicular to the beginning

of the path, and the profile always remains normal to the path as it is swept along its length.

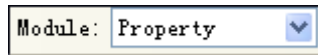
4 **Loft**  is to create a loft feature

You add a solid loft feature by creating two or more sections from selected edges and by defining one or more loft paths.

Tip:

- ✓ You can also import the 3D geometry model into the ABAQUS/CAE from other CAE or CAD software such as Solidworks. **File→Import→Part.**
- ✓ More details see *11. The Part module of ABAQUS/CAE User's Manual in ABAQUS Documentation*

3.3 Property Module



You can specify the properties of a part or part region by creating a section and assigning it to the part. In most cases, sections refer to materials that you have defined. Beam sections also refer to profiles that you have defined.

1. Create materials



A material definition is with its own name and is independent of any particular section. It specifies all the property data relevant to a material by including a set of material behaviors. you can refer to a single material in as many sections as necessary. ABAQUS/CAE assigns the properties of a material to a region of a part when you assign a section referring to that material to the region.

2. Create section



Sections are named and created independently of any particular region, part, or assembly. A section contains information about the properties of a part or a region of a part.

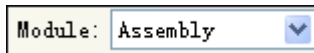
3. Assign section




You can assign section properties to a part by first creating a section and then selecting Assign Section to assign the section to a part or to a region of a part. Section properties that you assign to a part are assigned automatically to all instances of that part in the assembly.

Tip: More details see *12. The Property module of ABAQUS/CAE User's Manual in ABAQUS Documentation*

3.4 Assembly Module



When you create a part, it exists in its own coordinate system, independent of other parts in the model. In contrast, you use the Assembly module to create instances of your parts and to position the instances relative to each other in a global coordinate system, thus creating the assembly. You position part instances by sequentially applying position constraints that align selected faces, edges, or vertices or by applying simple translations and rotations.

Icon  is used to **create instance part**, which means the part in part module will appear in Assembly module in instance way.

An instance maintains its association with the original part. If the geometry of a part changes, ABAQUS/CAE automatically updates all instances of the part to reflect these changes. You cannot edit the geometry of a part instance directly.

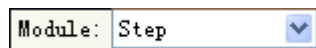
A model can contain many parts, and a part can be instanced many times in the assembly; however, a model contains only one assembly. Loads, boundary conditions, fields, and meshes are all applied to the assembly. Even if your model consists of only a single part, you must still create an assembly that consists of just a single instance of that part.

There are two types of instance part:

- ✓ Dependent instance part, for which the mesh will be operated on part.
- ✓ Independent instance part, for which the mesh will be operated on instance.

Tip: More details see *13. The Assembly module of ABAQUS/CAE User's Manual in ABAQUS Documentation*

3.5 Step Module



You can use the **Step Module** to perform the following tasks:

Create analysis steps



Within a model you define a sequence of one or more analysis steps. The step sequence provides a convenient way to capture changes in the loading and boundary conditions of the model, changes in the way parts of the model interact with each other, the removal or addition of parts, and any other changes that may occur in the model during the course of the analysis. In addition, steps allow you to change the

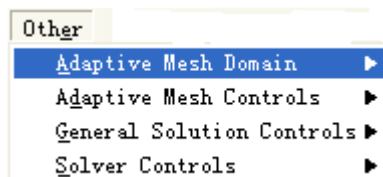
analysis procedure, the data output, and various controls. You can also use steps to define linear perturbation analyses about nonlinear base states.

Specify output requests

ABAQUS writes output from the analysis to the output database; you specify the output by creating output requests that are propagated to subsequent analysis steps. An output request defines which variables will be output during an analysis step, from which region of the model they will be output, and at what rate they will be output.

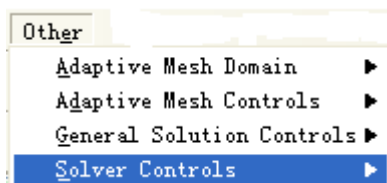
Output requests include **create field output**  and **create history output** .

Specify adaptive meshing



You can define adaptive mesh regions and specify controls for adaptive meshing in those regions.

Specify analysis controls



You can customize general solution controls and solver controls.

Tip: More details see *14. The Step module of ABAQUS/CAE User's Manual in ABAQUS Documentation*

3.5 Interaction Module: Interaction ▼

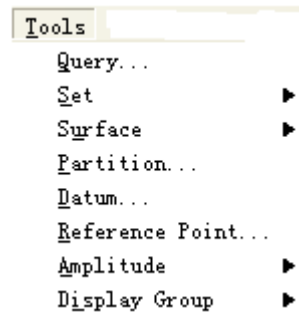
Interactions are step-dependent objects, which mean that when you define them, you must indicate in which steps of the analysis they are active.

The functions included in **Interaction Module** are

- ✧ Contact interactions.
- ✓ Elastic foundations.

- ✓ Thermal film conditions.
- ✓ Radiation to and from the ambient environment.
- ✓ A user-defined actuator/sensor interaction.
- ✧ Tie constraints.
- ✧ Rigid body constraints.
- ✧ Display body constraints.
- ✧ Coupling constraints.
- ✓ Shell-to-solid coupling constraints.
- ✧ Embedded region constraints.
- ✧ Equation constraints.
- ✓ Connectors.
- ✓ Inertia.
- ✓ Cracks.
- ✓ Springs and dashpots.

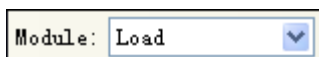
Tip: The **Set** and **Surface** toolsets in Interaction module allow you to define and name certain region of your model. After your initial defined set or surface, you can repeat using the region with the set or surface name rather select the region again.



ABAQUS/CAE does not recognize mechanical contact between part instances or regions of an assembly unless that contact is specified in the Interaction module.

Tip: More details see 15. *The Interaction module of ABAQUS/CAE User's Manual in ABAQUS Documentation*

3.6 Load Module



You use the **Load module** to define and manage the following prescribed boundary conditions:

- ✓ Loads
- ✓ Boundary conditions
- ✓ Fields
- ✓ Load cases

You can use the load, boundary condition, and field managers to view and manipulate the stepwise history of prescribed conditions. You can also use the **Step** list located under the toolbar to specify the steps in which new loads, boundary conditions, and

fields become active by default.

Tip: More details see *16. The Load module of ABAQUS/CAE User's Manual in ABAQUS Documentation*

3.7 Mesh

Mesh module allows you to generate meshes on parts and assemblies created within ABAQUS/CAE.

You can create a mesh according to the following process.



Seed Part Instance: Specify mesh density.



Assign mesh control: Specify element shape as well as the different mesh techniques to generate meshes.



Assign element type: Specify element type according to requirement.



Mesh part or region as well as delete mesh part or region.



Verify mesh: provide you with information concerning the quality of the elements used in a mesh.

Tip: you can use partition toolset to partition complex models into simpler subregions.

Then refine mesh your interested region, such as the crack region. In addition, **mesh edit** can edit the existed mesh.



Tip: More details see *17. The Mesh module of ABAQUS/CAE User's Manual in ABAQUS Documentation*