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Undo and Redo

With **Undo/Redo**, you can easily reverse any mistake in CoCreate Modeling.

In the **Main Task bar**, you can undo or redo one step at a time. Click **Undo One** or **Redo One**.

To undo or redo,

1. Click **Edit** > **Undo/Redo**. The **Undo** dialog opens.
2. To undo steps, select **Backward**. Either click **Next**, or type the number of steps you want to undo. Click **OK**.
3. To redo steps, select **Forward**. Either click **Next**, or type the number of steps you want to redo. Click **OK**.
4. If you want to see the history of all the commands you've entered in your CoCreate Modeling session, check **Expand**. The **Undo** dialog expands to display:
   - **States**: lists the commands you've entered in your CoCreate Modeling session.
   - **History**:
     - **Max Back**: displays the maximum number of commands you can undo.
     - **Max Forward**: displays the maximum number of commands you can redo.
   - **Limit**: displays the number of steps to keep in the **Undo/Redo History**. To set or change the limit, check **Steps**, and type a number in the field.
5. Click OK to complete the operation..

CoCreate Modeling tracks both the command history of CoCreate Modeling and any active module applications.

- When you select **Undo** from CoCreate Modeling, you may get a warning that you're about to undo a module application command.
- When you select **Undo** from a module application, you may get a warning that you're about to undo a CoCreate Modeling command.

Either continue with the **Undo**, or click **Abort**.

Compare parts

With **Compare Parts**, you can compare the geometry of two nearly identical parts.

This option helps you compare different versions of a model. For example, a company might construct a part and send it to a partner for finalization. The partner modifies the part and sends it back. With **Compare Parts**, the differences between the first and second versions can be analyzed, displayed, and kept for further modeling operations.

CoCreate Modeling highlights the differences:

To compare parts,

1. Load both parts you want to compare in the viewport.
2. Click **Menu 2** in **Part & Assy**.
3. Click **Compare Parts** in the **Checks** section.
4. Select the original or **Reference** part in the Structure Browser.
5. Select the same face on the changed or **Check** part.
6. If you want, you can define the accuracy to be used for the part comparison. The lowest resolution of both parts is selected by default. You can choose a lower resolution if you want a rougher comparison of the parts.
7. CoCreate Modeling needs to know how to match the parts so they must be aligned in the same position. Choose an alignment option:
   - **No Align**: The parts are already aligned.
   - **Auto Align**: CoCreate Modeling positions the **Check** part so the selected faces of the **Reference** part and **Check** part are aligned. If more than one alignment is possible, you can choose from a list of available alignments using the **Next** and **Previous** buttons.
   - **Auto Align with Vertices**: If an automatic alignment is not possible, you can select a vertex on the selected **Reference** part face and the matching vertex on the **Check** part face. CoCreate Modeling positions the **Check** part so the two vertices are aligned.
   - **Manual Align**: If automatic alignment (with or without vertices) is not possible, you can position the **Check** part using the dynamic position dialog. You can also use the **Mate Align** options for manual alignment.
8. Click **Calculate**. CoCreate Modeling displays the **Reference** part and the **Check** part in separate viewports. The differing faces, edges, and vertices are highlighted in different colors.

Limitations

- Compare parts only compares geometric and topologic information. It doesn't include any other information such as blends or other feature data in the comparison.
- Scaled parts may not compare accurately. The farther geometry is from the reference points (the points at which the parts align), the more likely the comparison will fail and produce **Only Ref** or **Only Check** results.
- The comparison method for edges and faces is based on a smart sampling technique. Because it samples a finite (but representative) number of points, the distance returned by Advanced Analysis is an approximation. More information...

Related topics

View part comparison results

View part comparison results

To view and manipulate the comparison results,

1. Select **Synchronize Viewports** so the two parts will move together as you rotate, zoom, etc.
2. Select which geometry is compared in the **Show Types** section: **Faces**, **Edges**, or **Vertices**.
3. In the **Show Matches** section, select which matches you want to display:
   - **Only Ref**: Highlight the differences that exist only in the reference part.
   - **Only Check**: Highlight the differences that exist only in the check part.
   - **DifferentGeo**: Highlight differences in geometry.
   - **AffectedGeo**: Highlight geometry that matches, but is affected either by differences between the parts or differences in geometry type.
4. Expand the **Report Matching Faces** section to view a summary of the types of differences found.
5. Click **Highlight Matching Faces** and select a face, edge or vertex. CoCreate Modeling will highlight the match on the other part. The matching may not be unique. For example, a face on one part might be split into two faces on the other part.
6. Expand **Advanced Analysis** and select the parts' corresponding faces for a detailed comparison of faces, edges or vertices.

   You can click **Change Definition** at the top of the dialog to return to the previous dialog and change the comparison.
7. You may want to document the specific changes to the part. In the **Keep Results** section, you can check **Faces**, or **Edges** and an assembly *Compare_Parts* is created when you click **OK**. Any matching face or edge is copied and grouped into parts with regard to the **Show Types** and the owner of the faces or edges (**Reference** and **Check** part). **Faces** is selected by default.

   No part is created if no face or edge exists for certain **Show Types**.
8. If you do not want to keep your results, uncheck both boxes and click **OK**.

Related topics

Compare parts

Integrate imported models into CoCreate Modeling

Imported models are models that were either created or modified in another CAD system.

Companies don't always model every component of their products internally; they often send models out for revision, or they routinely receive models from subcontractors who may not use CoCreate Modeling or the same CoCreate Manager Server database. This situation can disrupt the product design process. Why?

Different CAD systems:

Any time you load an imported model into CoCreate Modeling, CoCreate Modeling views it as a new model, and it assigns it a new Model ID.

- When you send models out to subcontractors, the revisions the subcontractor returns lose the attributes and properties of your original models. You also lose any revision history.
- When you routinely receive models from subcontractors, the first model revision receives a new Model ID, as expected. However, CoCreate Modeling interprets any subsequent revision of that model as new models so you lose any revision history.

Different CoCreate Manager Server database: If your subcontractor doesn't use the same database, the models retain the same Model IDs, but they acquire different database attributes. When you save these models, your database interprets the revisions as new models rather than as new revisions.

Use the **Versioning** tools in the **Parts and Assemblies** menu to successfully integrate parts or assemblies into CoCreate Modeling and your CoCreate Manager Server database.

- To integrate parts or assemblies from another CAD system,
- To integrate parts or assemblies from another database,

Integration hints

- When you integrate, the original part or assembly takes the position of the imported model. If you want to keep the position of the original, you'll need to reposition the imported model before reintegration.
- If your subcontractor has mistakenly deleted a part or assembly, you can share it from the reference into the imported assembly.
- If your subcontractor has created a part or assembly you don't want, just delete it from the imported assembly.

Limitations

Reintegration matches parts and assemblies between models, but not specific faces, edges, or vertices. Therefore,

- **Labels**, **Docuplanes**, **Relations**, or **Features** lose their points of reference during reintegration. If you want to keep this data, you'll have to modify or recreate it before reintegration.
- **Viewsets**, **Relations**, **Animations**, or **Clash Analyses** you can transfer manually after reintegration.